Circuit Design Using Simulation and Virtual Instrumentation

An Introduction

Applications in Biomedical Engineering

Patrick Noonan
Business Development Mgr
National Instruments
Electronics Workbench Group

Agenda

- **SPICE Overview and Simulation Tools Today**
- **Using Simulation and Virtual Instrumentation**

  Demonstrations Highlighted Using Multisim and LabVIEW

- **Design Case Studies: Biomedical Circuits and Applications**
  - Sensor Emulation for ECG Amplifier Design
  - H Bridge Motor Simulation for Medical Pump Design
  - Design of a Uniform Light Source
  - Sigma Delta ADC Development
  - Video Signal Generation with Video Amplifier Design in SPICE
  - Measurements and Automating Design Validation and Testing

Engineers: Roles, Tasks and Risks

- Challenge for Design Engineers:
  - **Develop a "widget" quickly, inexpensively and make sure it works right.**

- Approach 1 – Trust the app notes, datasheets, build it and then test it.
  - Risk: No innovation – typically a cookie cutter approach
  - Risk: Uh-oh (some assumptions were wrong, troubleshoot it and possibly go back to the drawing board)

- Approach 2 – Simulate the heck out of it, see that it works, build it and test it
  - Risk: Project delayed as you try to find or develop models
  - Risk: Uh-oh (assumptions were wrong in the model – design doesn't work or project takes too long

- Apprach 3 – Simulates pieces (prototype those that are riskier), build it and test it
  - Risk: Ok, you can never get rid of risk.
  - But… You can minimize the risks in Approach 1 & 2
  - Design will take longer than Approach 1 however design is more likely to be close to spec the first time

Use **SPICE Analysis and Measurements** validate designs to **REDUCE Risks**

Electrical Engineering Design Tools

- **SPICE and Circuit Analysis**
  - Part of most modern day circuit design tools – analog, digital and mixed signal
  - Sometimes simulation is not as 'integrated' – outside the normal design flow
  - Some tools are more user intuitive than others
  - Some tools make SPICE overcomplicated (thus limiting its use)
  - Can be very useful if tools simplify its use **AND** fit it into design flow

- **Virtual Instrumentation**
  - Using the PC to perform measurements, calculations and analysis for testing
  - Allows flexibility of adding customization and integration of many measurement devices into a single application
  - Allows for Automation!
  - Generally NOT used within context of SPICE

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 circuit 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.
  - RF with Electromagnetic Field Solvers (Agilent Advanced Design System™ or Ansoft Designer ®)

SPICE History of Circuit Simulation

- **SPICE**
  - Developed as part of Thesis paper at University of California at Berkeley by Larry Nagle
- **History**
  - 1969 – CANCER (Computer Analysis of Nonlinear Circuits Excluding Radiation)
  - 1972 – SPICE 1
  - 1975 – SPICE 2
  - 1985 – SPICE 3
  - 1993 – SPICE 3F4
- **Popular Commercial Versions**
  - OrCAD® PSpice®
  - LTspice/SwitcherCAD™ III
  - Multisim™
  - TINA™ by DesignSoft

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

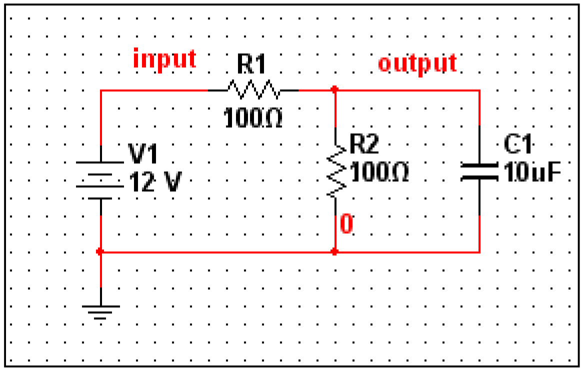
- **Example SPICE netlist**

  R1 input output 100

  R2 output 0 100

  C1 output 0 0.00001

  V1 0 input 12



- **Subcircuit SPICE models**
  - Combination of lower order primitive models to reflect behavior and performance of a component
  - Command ".subckt" describes start of model
  - Command ".ends" encloses end of circuit
  - Example shown for a Bipolar Junction Transistor

```
.SUBCKT BJTEXAMP 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

.ends BJTEXAMP

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

Advantages to Using SPICE with Virtual Instrumentation

Mathematical capabilities of **SPICE** to accurately model complex circuits and devices

- AND –

Measurement capabilities of **Virtual Instrumentation** (such as data collection, automation, testing, etc)

SPICE

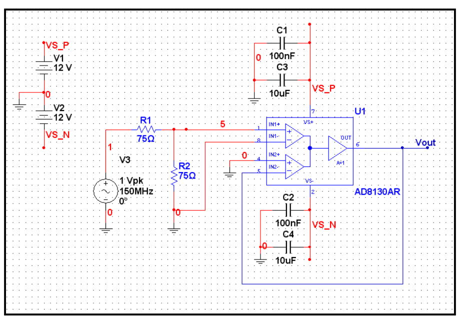
Schematic, Simulation, Analysis

Virtual Prototype

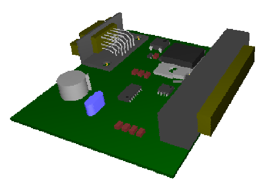
Testing

VI Software

Virtual Measurements



Comparison between simulation data and measurements is simplified

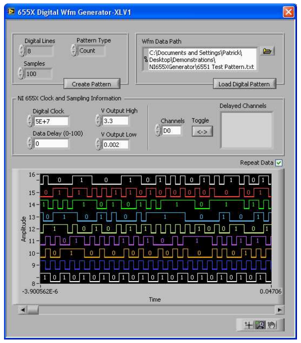


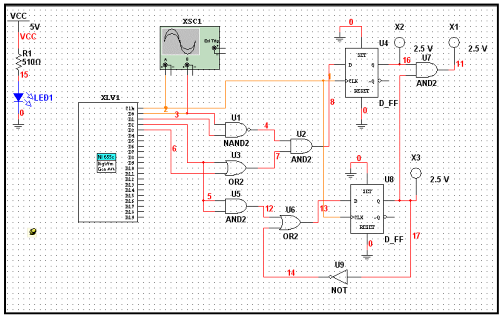
Measurements

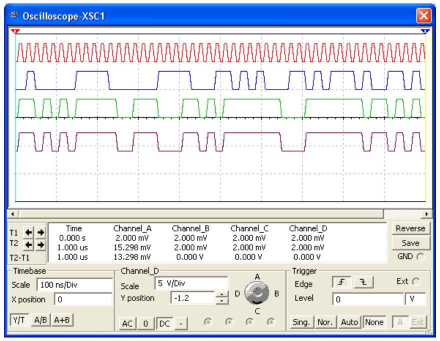
Physical Measurements

Introducing Multisim and LabVIEW for Circuit Design

- Link between SPICE simulation tool and Virtual Instrumentation tool
- Many Engineering Circuit and System Design Possibilities Open Up
  - Making more than traditional V & I Measurements directly within SPICE
  - Sensor Emulation
  - Direct Link between Simulation and Measurement Data
  - Tie between SPICE into Test Hardware
  - System Level and Algorithm Prototyping
  - Design Automation and Optimization

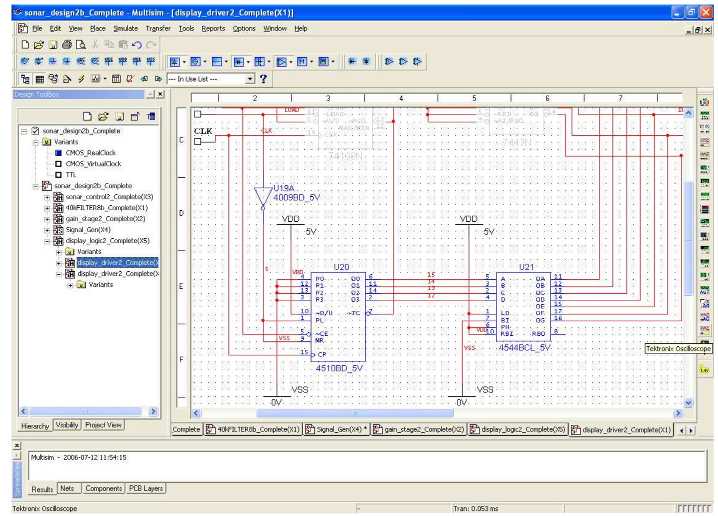


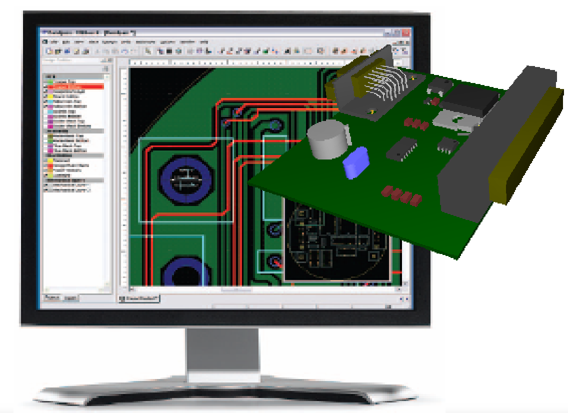




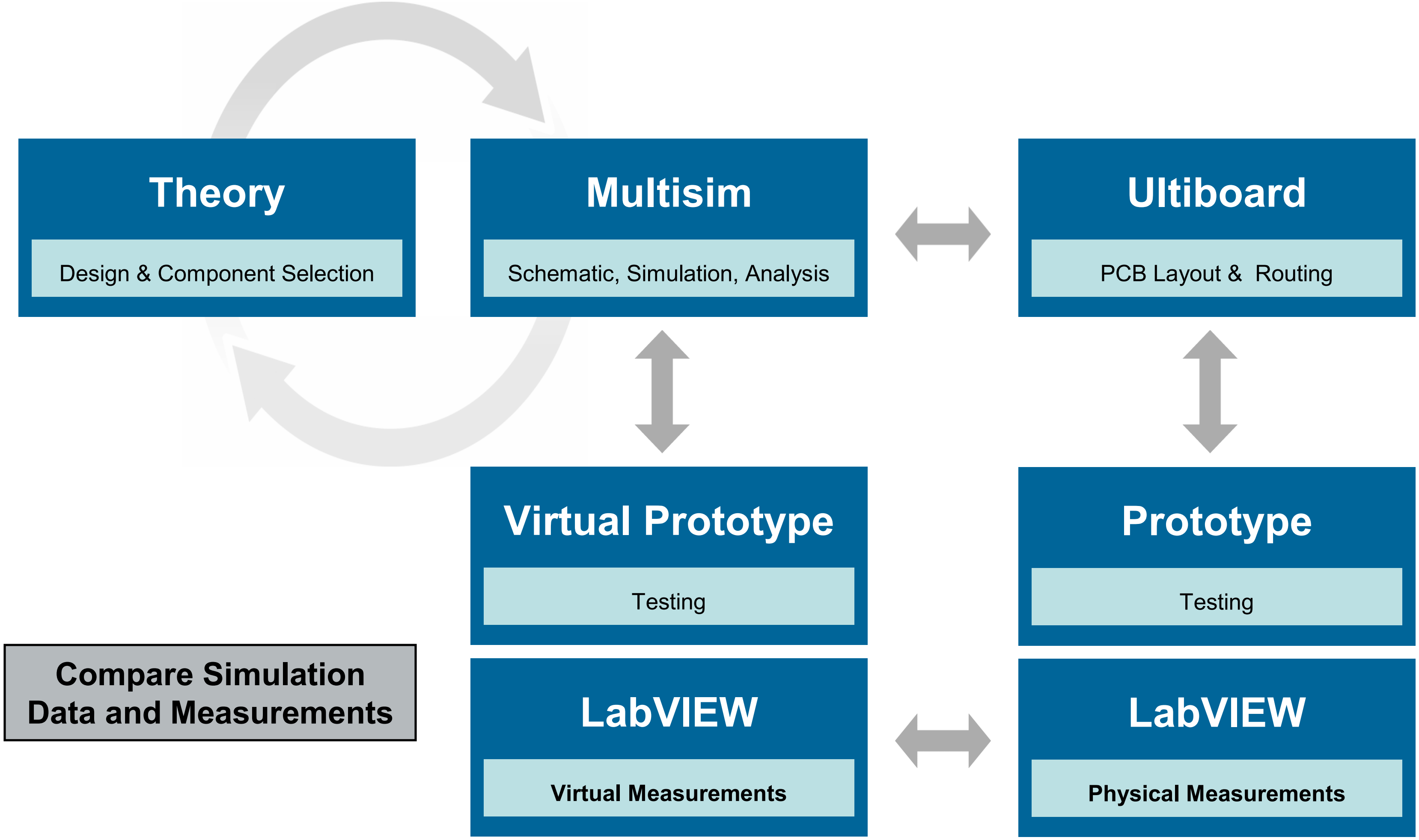
NI Multisim – Schematic Capture, Simulation and Analysis

- Graphical based schematic capture and integrated SPICE simulation
  - Digital and Analog Co-simulation
- Thousands of components immediately ready for simulation
  - Place symbol onto schematic and click the Simulate Button
- Create custom components and models
- Virtual Instruments for immediate testing
- Advanced analyses for design validation
- Integration with Ultiboard and other PCB tools for Prototyping and Full PCB Layout



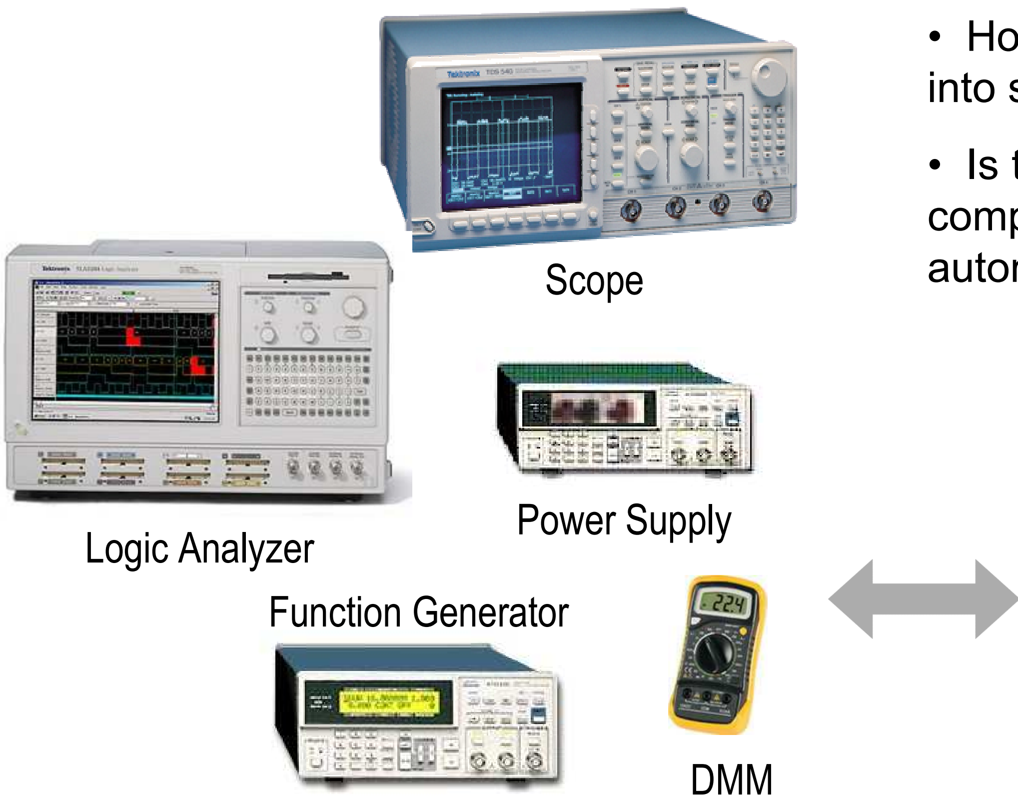


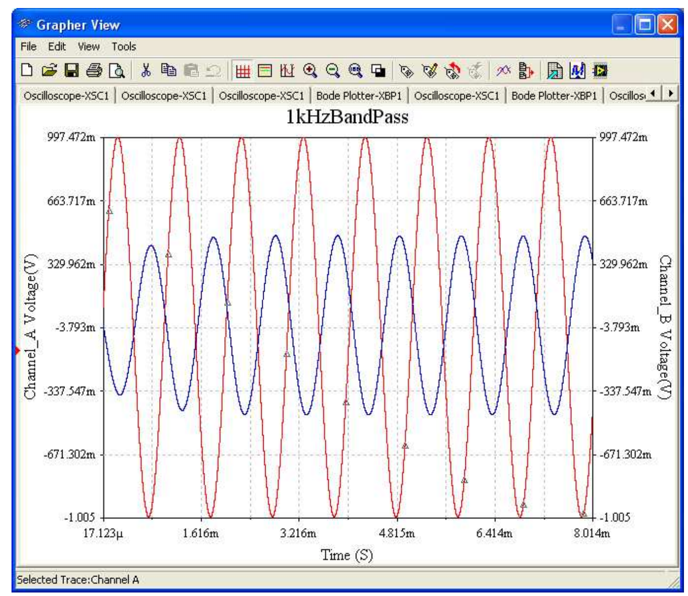
Integrated Design and Test Flow – Multisim, LabVIEW and Ultiboard



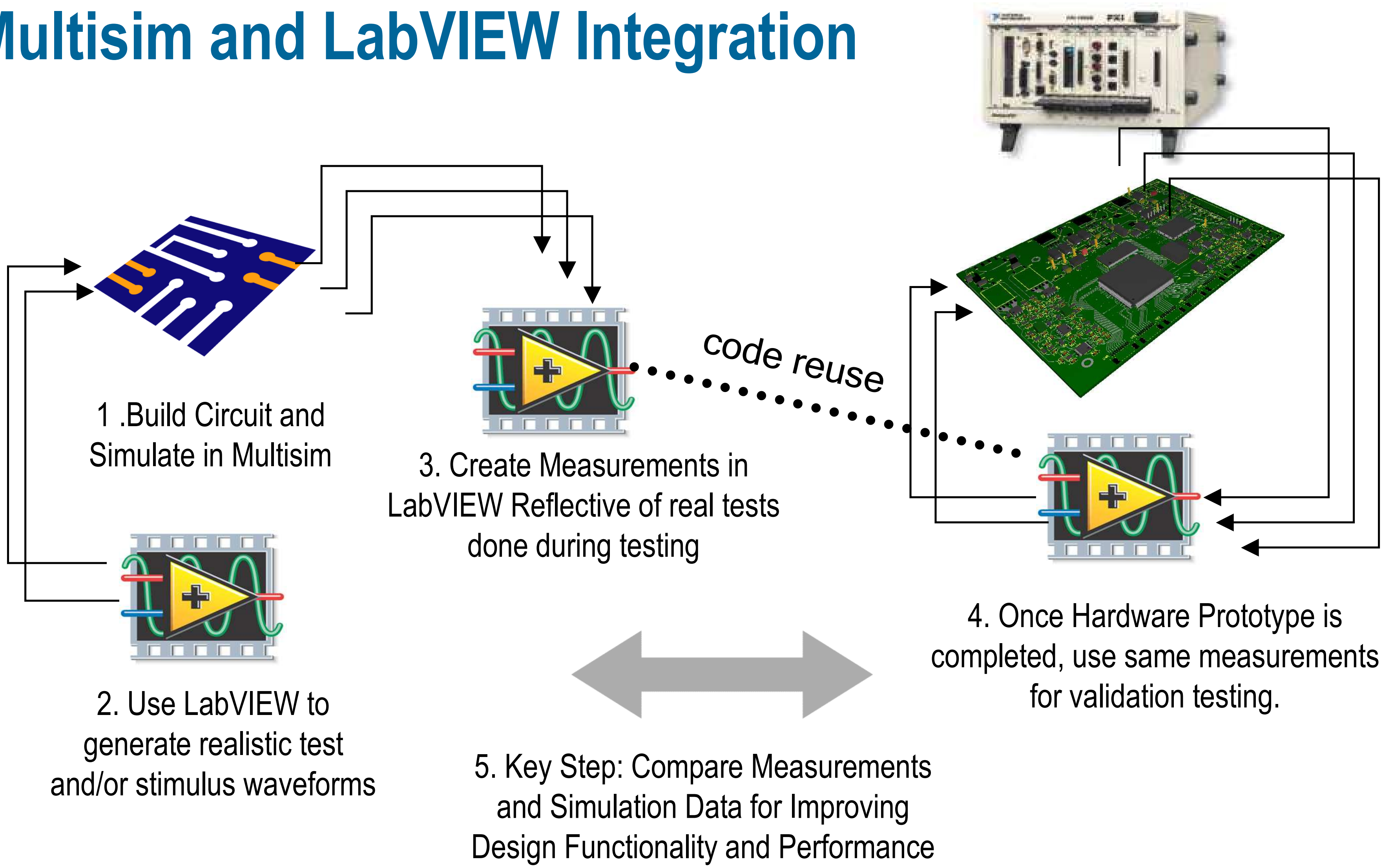
Simulation and Measurements for Design Engineers

- How do you effectively compare test bench data with simulation data?
- How can you bring in measurement data into simulation?
- Is there anyway to perform simulations, compare results and optimize the design automatically?

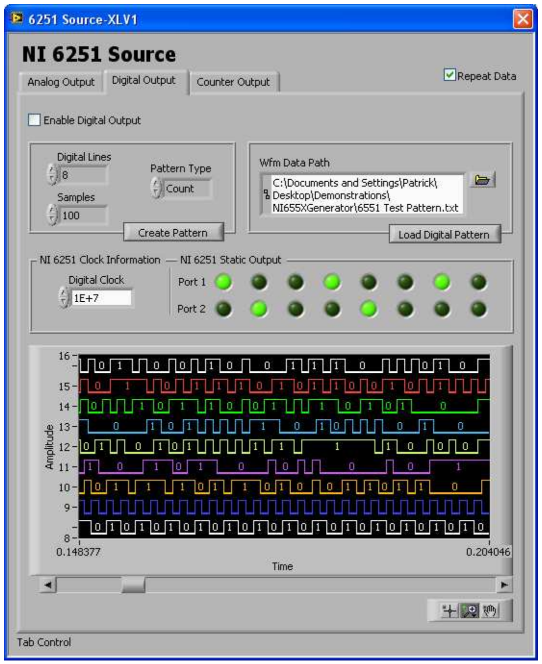


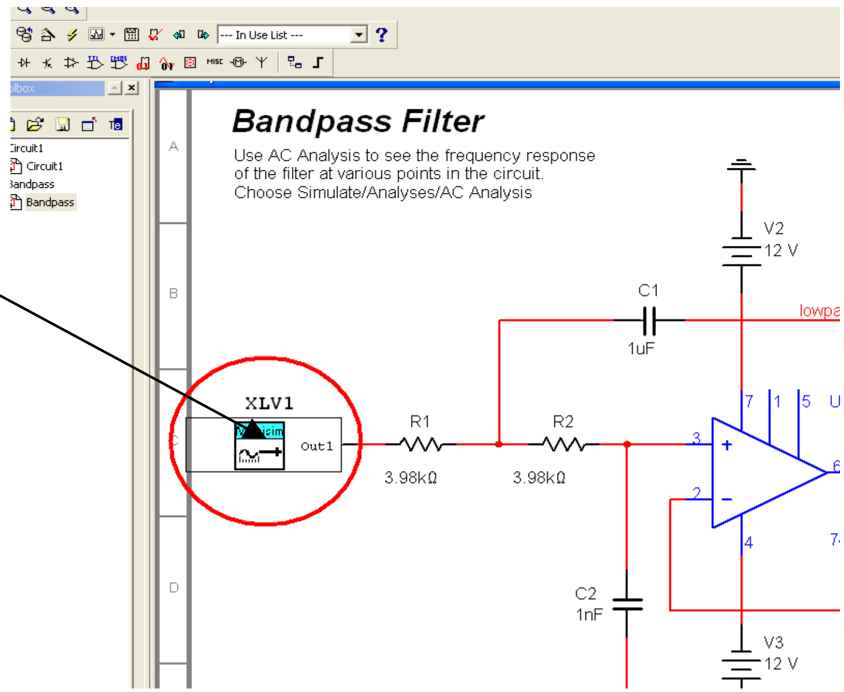


Multisim and LabVIEW Integration



Test Capabilities in SPICE – LabVIEW Instruments





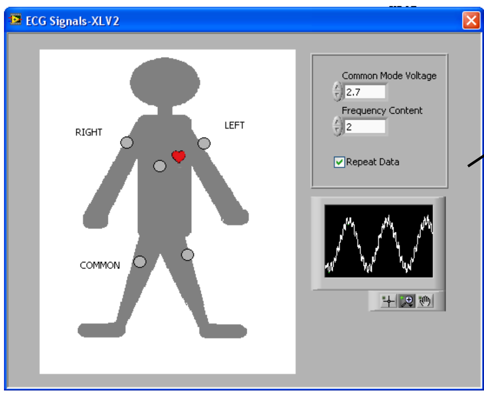
Example showing injection of real hardware test signal into circuit simulation using Virtual Instrument; Exact test pattern can be used on hardware prototype

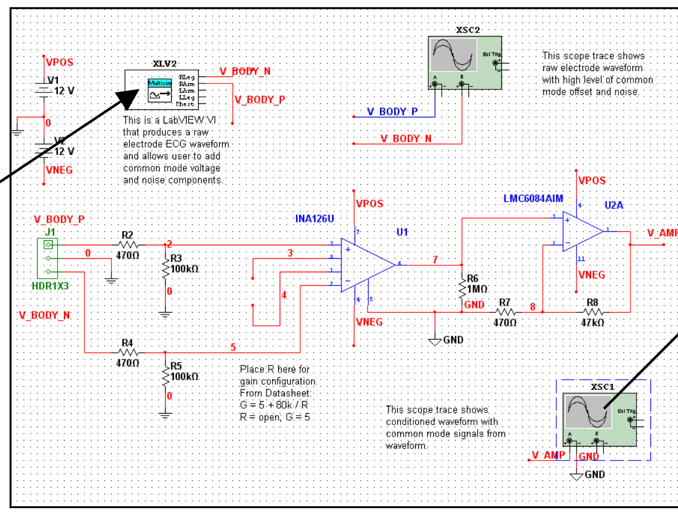
SPICE and Virtual Instrumentation Examples
Biomedical Engineering

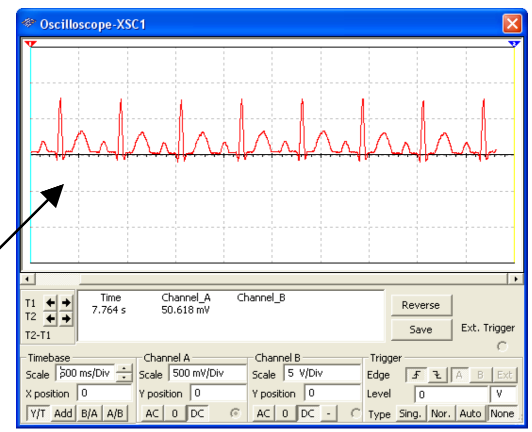
To download circuit files and associated Virtual Instruments, please go to www.ni.com/multisim

Example for ECG Amplifier Development

- ECG is typically a 1mV – 3mV waveform
- ECG Signal is typically riding on high Common Mode Component (2 to 3V)
- Due to the high output impedance high levels of noise are evident on electrodes
- **EXAMPLE:** In LabVIEW we can Prototype the Waveform (Referenced from the Electrode) and use controls to adjust Common Mode and Noise Component levels. Multisim can then be used to effectively design the ECG Amplifier to extract ECG Waveform.







1. LabVIEW constructs **Stimulus Waveform**

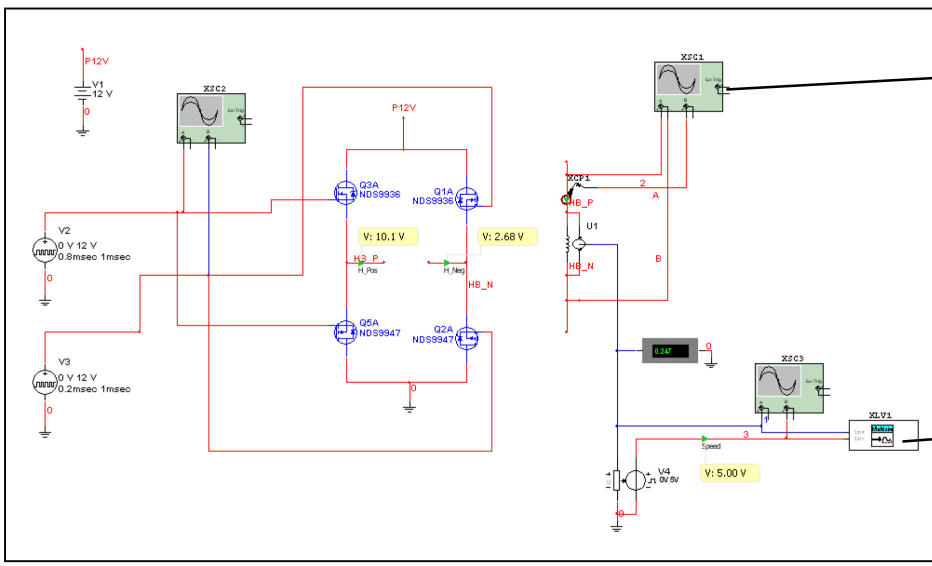
2. Multisim showing ECG Signal Extraction in Simulation

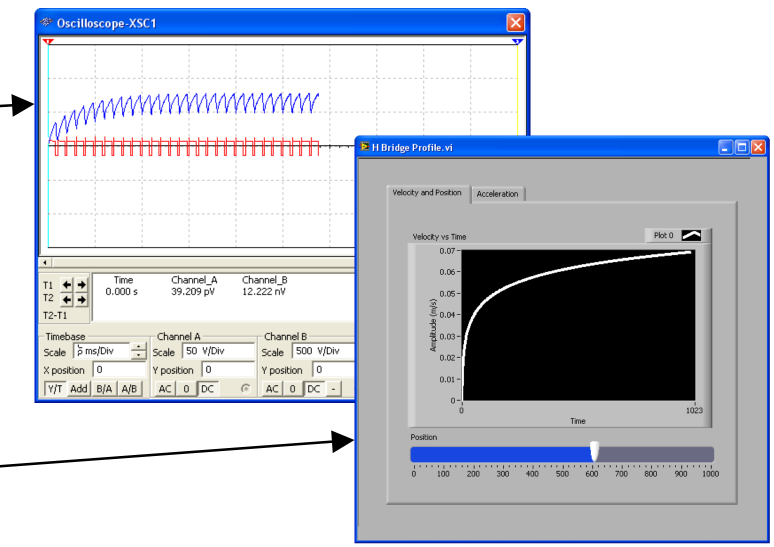
H Bridge Motor Simulation for Medical Pump Design

- Example Design showing H Bridge Drive Circuit Using Power Mosfets
- Mosfets gates controlled by PWM signals to control current through the motor

- **EXAMPLE:** In LabVIEW we are measuring the encoder signal from this H bridge / motor simulation and calculating Acceleration, Velocity and Position.

1. H Bridge Motor Simulation using SPICE Models for Power Mosfets, Motor, and Encoder



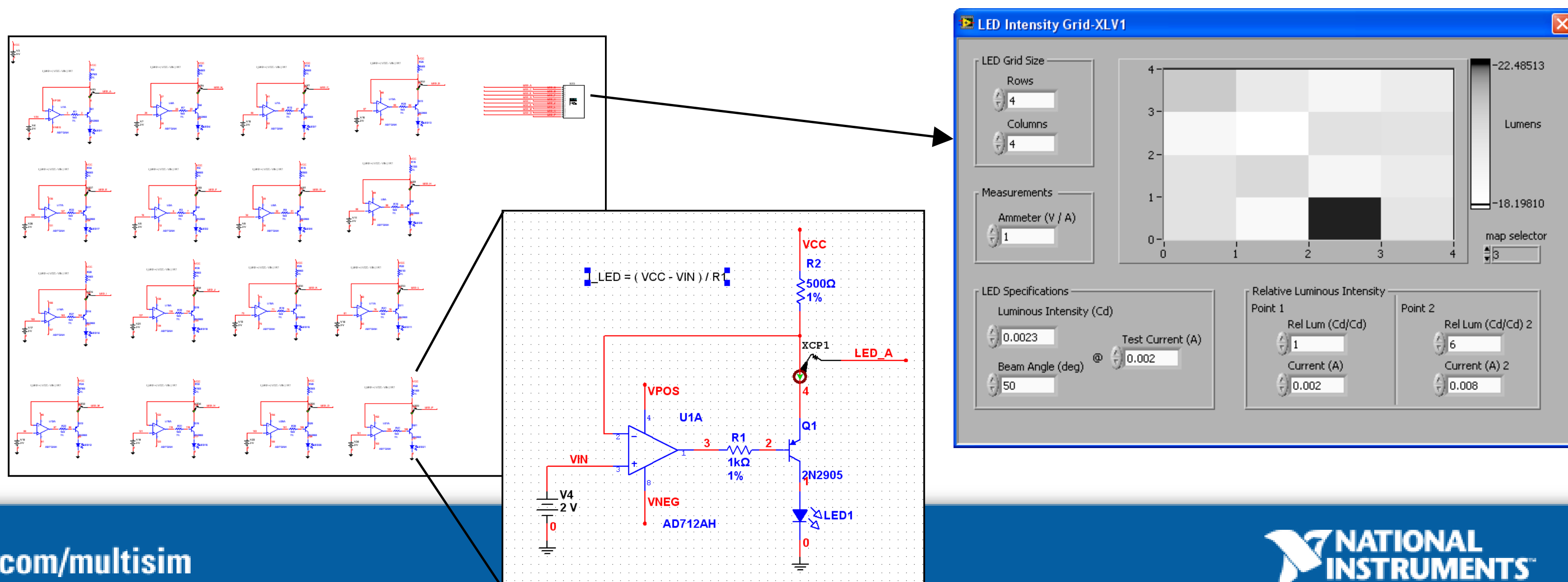


2. LabVIEW used to Measure Acc, Vel and Pos.

Design of a Uniform Light Source

- Uniform LED Array Design for Illumination
- Using Circuit in Multisim and LED Specification Data Built into LabVIEW
- Varying the Circuit Tolerances Yields Variances in Intensity Display Graph

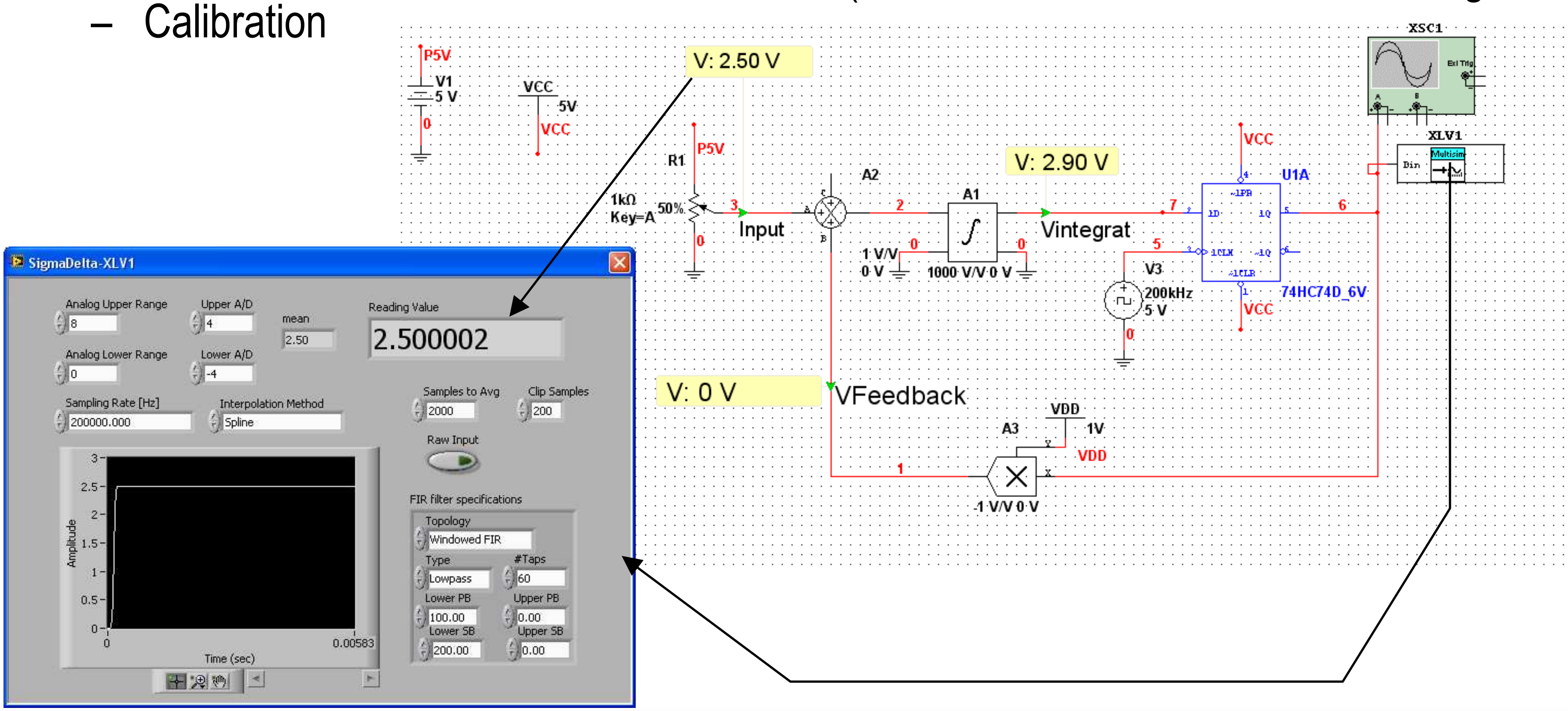
- **EXAMPLE:** In Multisim we can prototype a LED Array as a light source and Use LabVIEW to take derived Electo-Optical Measurements based on Simulation Data in Multisim and Specification Data from the LED data sheets



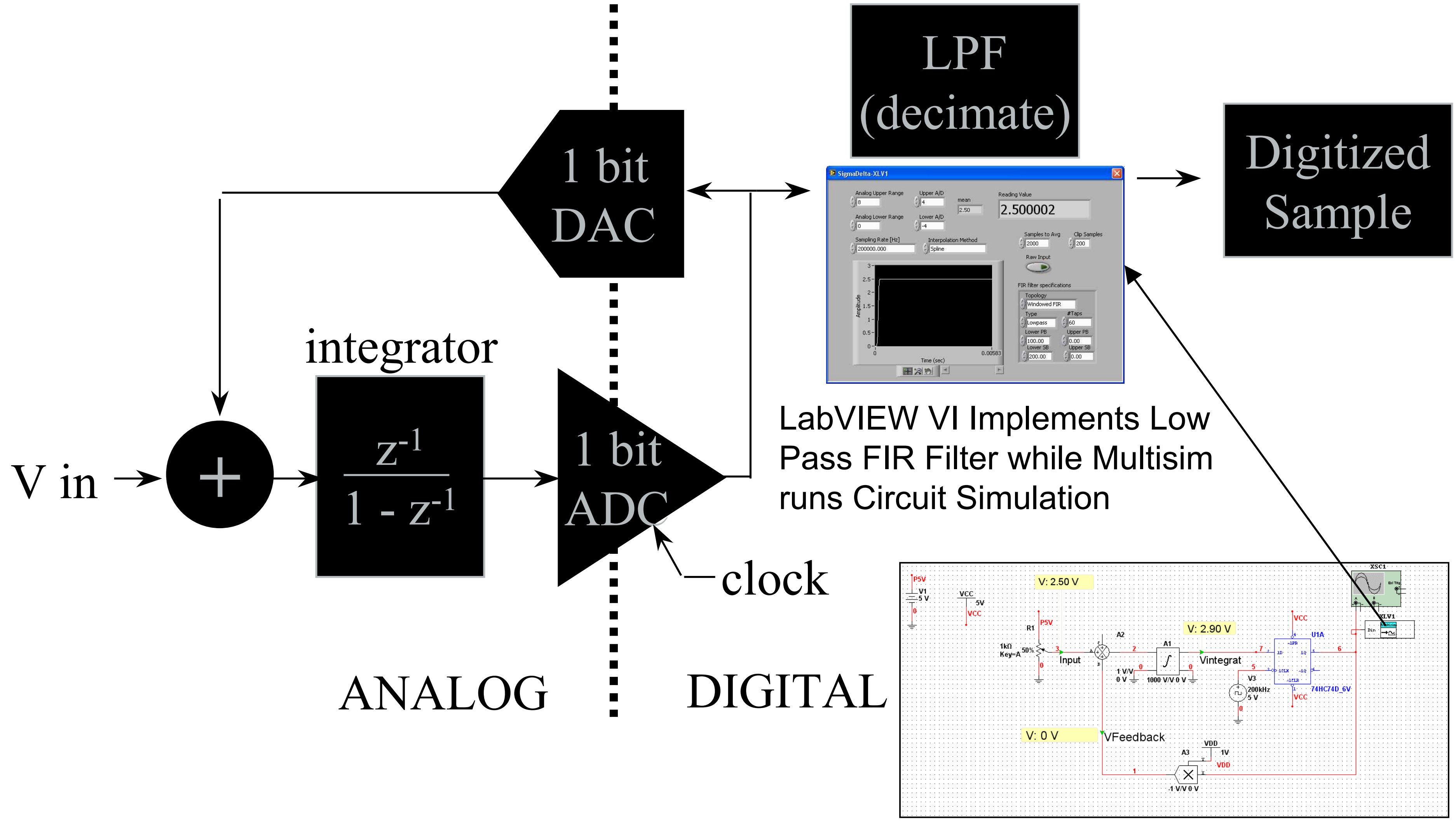
LabVIEW for Sigma Delta Circuit Development

- Multisim for **Sigma Delta** Circuit Construction
- **Digital Signal Processing** in LabVIEW
  - Digital FIR Filtering
  - Calibration

Example – Ideal 1st Order Sigma Delta ADC
(Used to Test LabVIEW DSP Filter Algorithm)

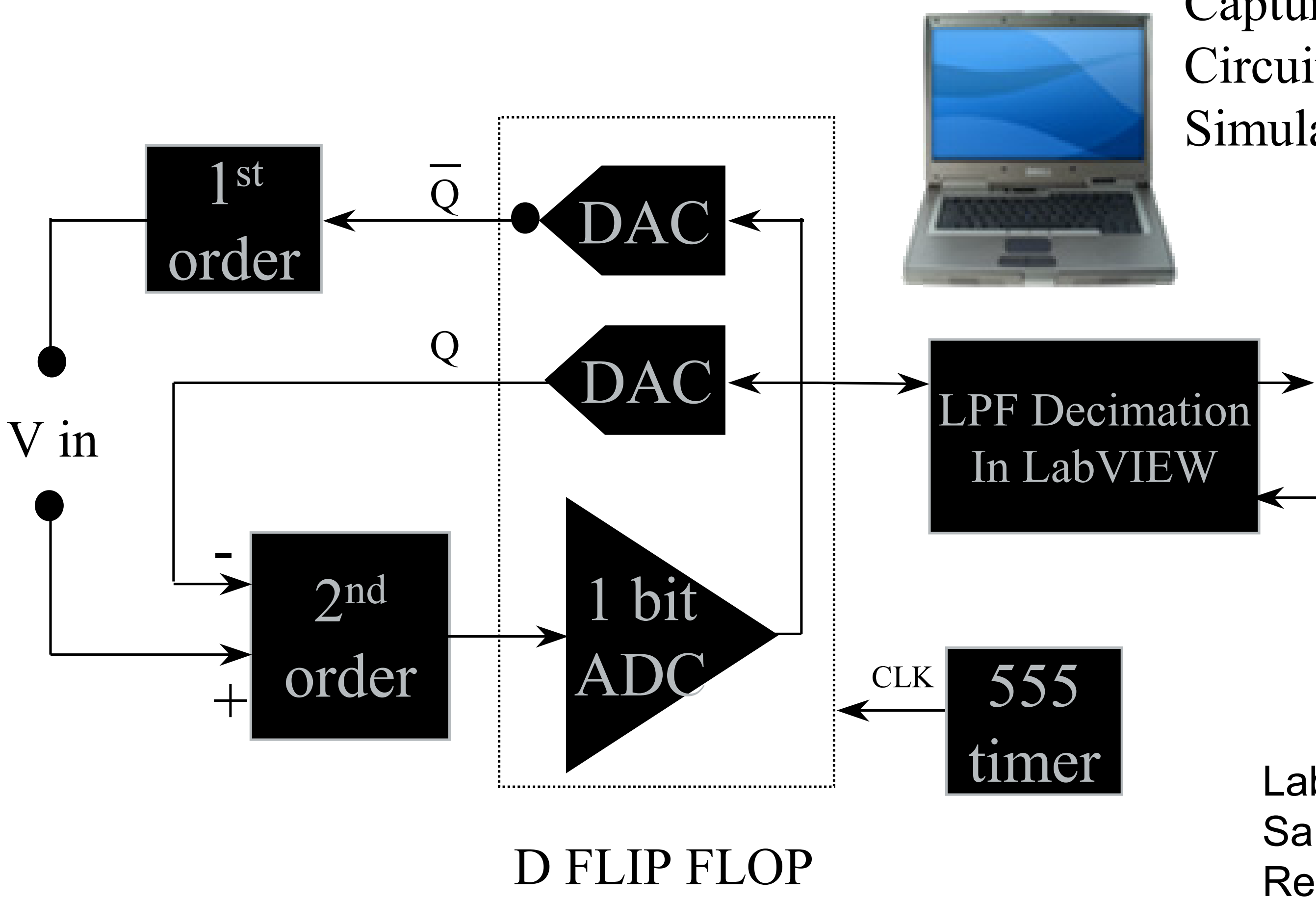


Step 1. Build Ideal 1st Order Sigma Delta Architecture

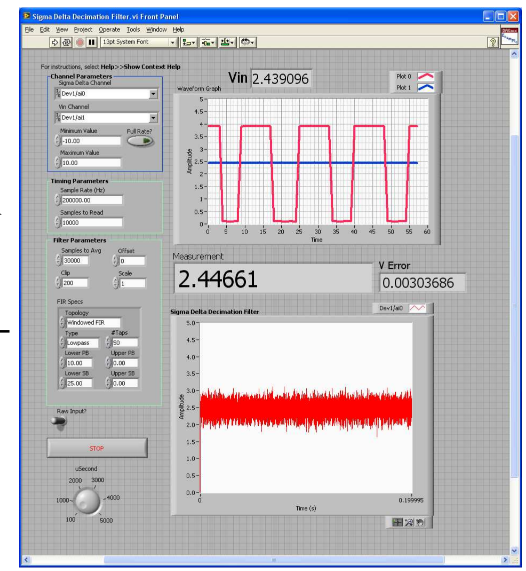


LabVIEW VI Implements Low Pass FIR Filter while Multisim runs Circuit Simulation

Step 2. Construct 2nd Order Sigma Delta Simulation and Build Prototype



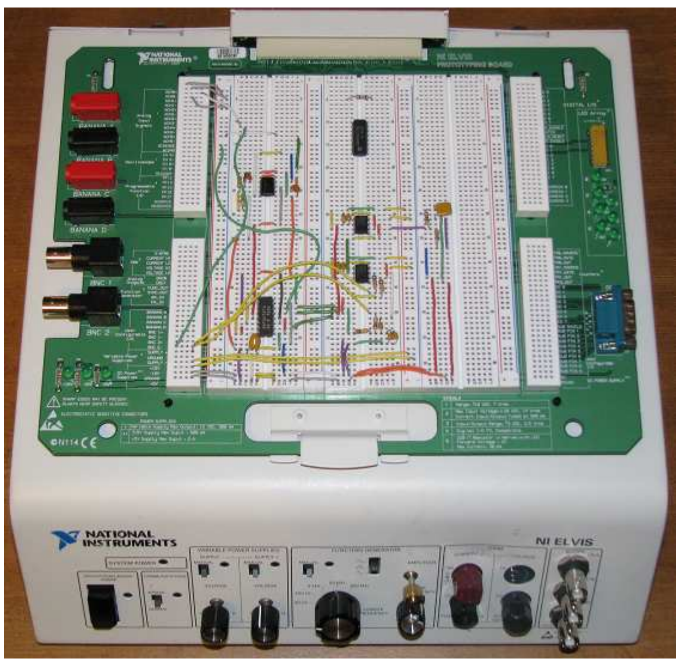
DAQ Card in Laptop Used to Capture Pulse Train from Circuit and Compare Simulation to Measured Data



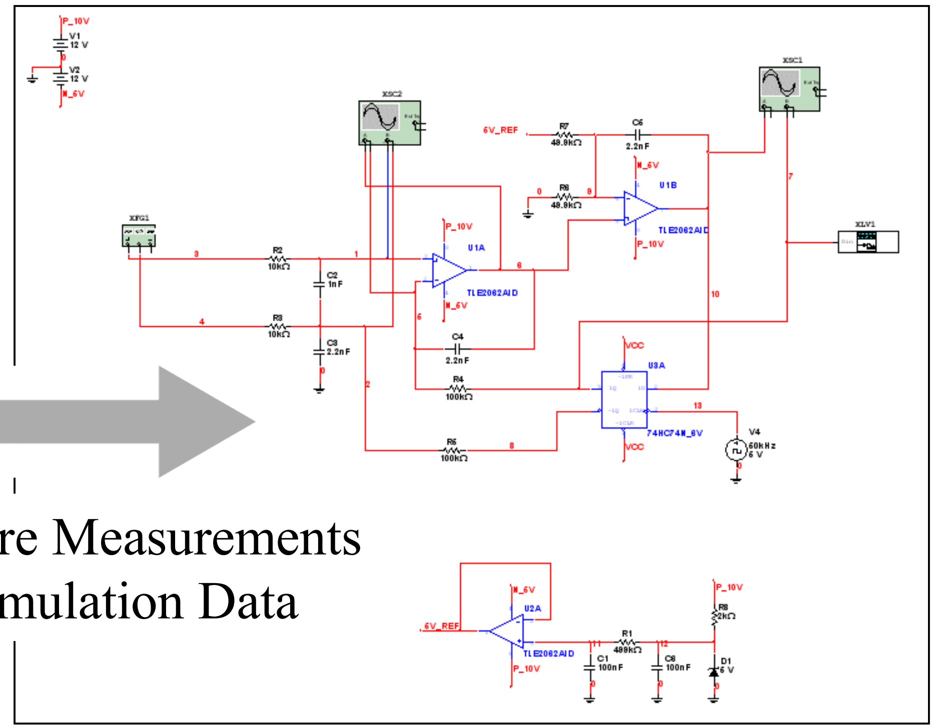
LabVIEW VI Implements Same Low Pass FIR Filter in Real Circuit Implementation

Step 3. Sigma Delta Simulation and Measurements

- Multisim used to create 2nd Order Sigma Delta circuit and simulation
- LabVIEW to build prototype decimation filter (LPF) for simulation
- NI-ELVIS to build circuit prototype
- DAQ to record waveform and LabVIEW to build same decimation filter with HW

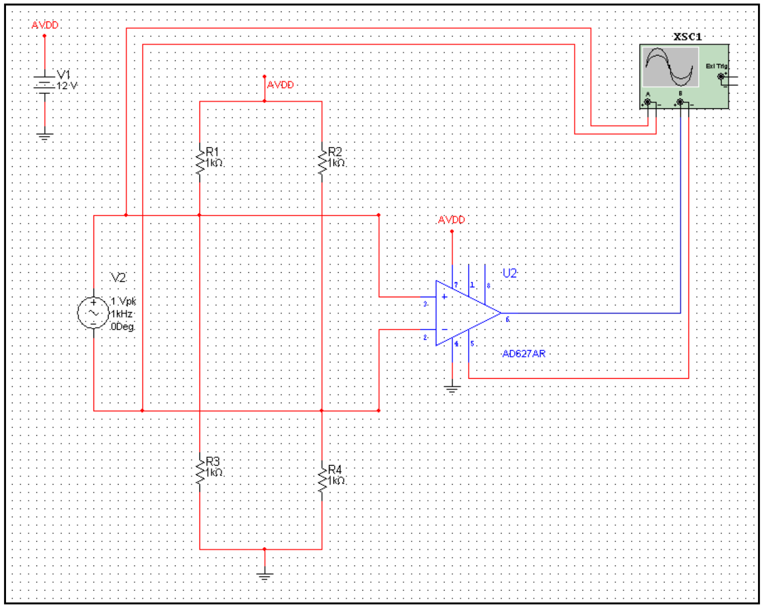


Compare Measurements with Simulation Data



Possible Next Step: Build Prototype PCB and Implement DSP Filter in LabVIEW Embedded

Video Amplifier: Basic High Speed Differential Amp



Video Test Signal Requirements:

RS-170

525 lines/frame

Line Frequency 15.735 kHz

Line Duration: 63.556 msec

Active Pixels / line: 640

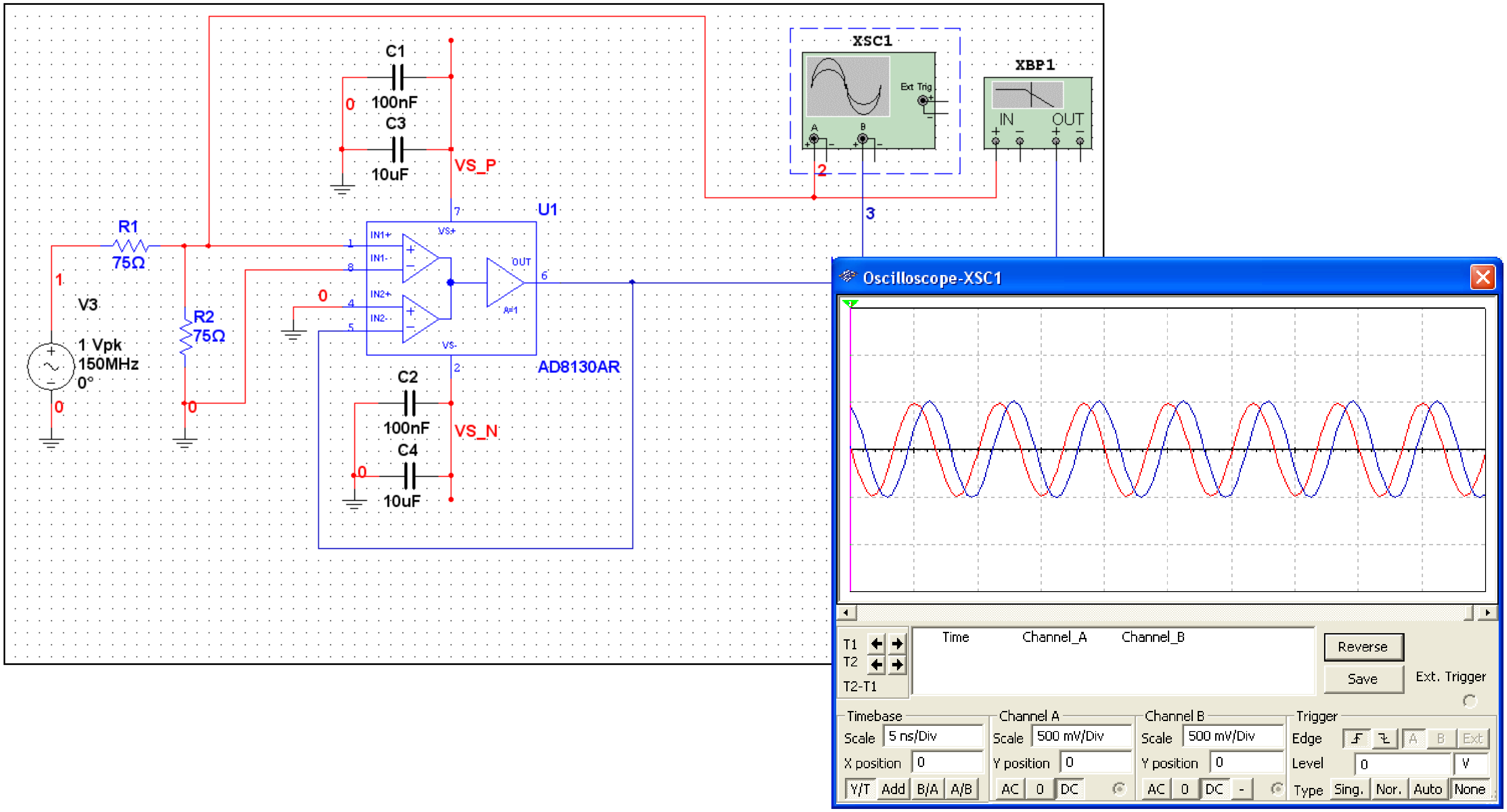
Pixel Clock ->

640 pixels/line / 52.66E-6 sec/line = 12.15 Mhz

Therefore need a high BW amplifier to ensure we are not impacting pixel data before digitization.

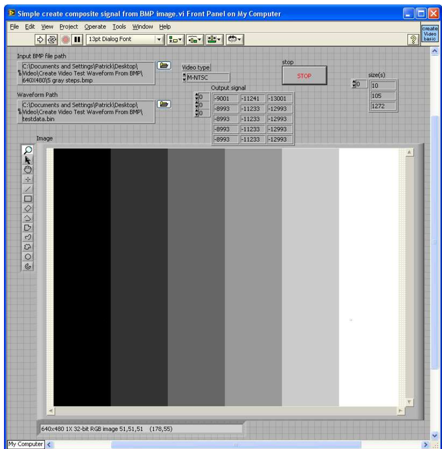
Specialized Differential Video Amp

Using Multisim to test AD8130 performance (magnitude – phase vs. frequency response)

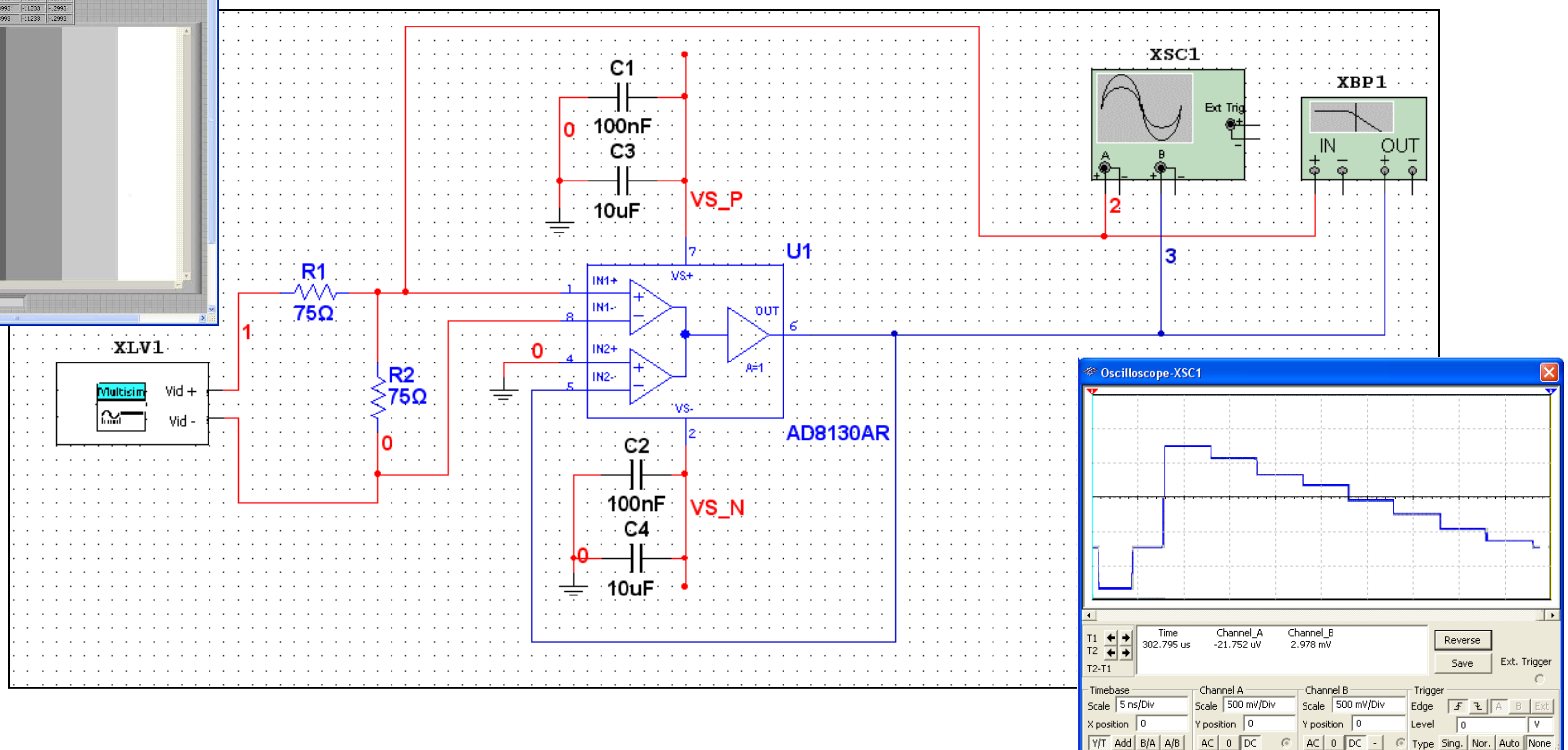


Specialized Differential Video Amp: LabVIEW Testing

NI LabVIEW to create Video Waveform Source from Video Test Pattern



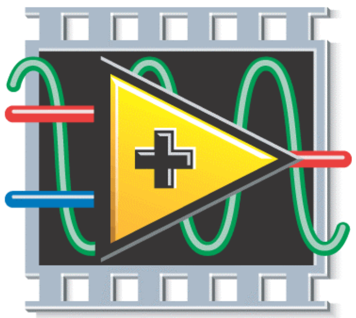
1. Use LabVIEW to Create a Video Test Image (BMP) to be created into Video test waveform.



2. Multisim Scope displays video pattern as simulation runs

LabVIEW Toolkits

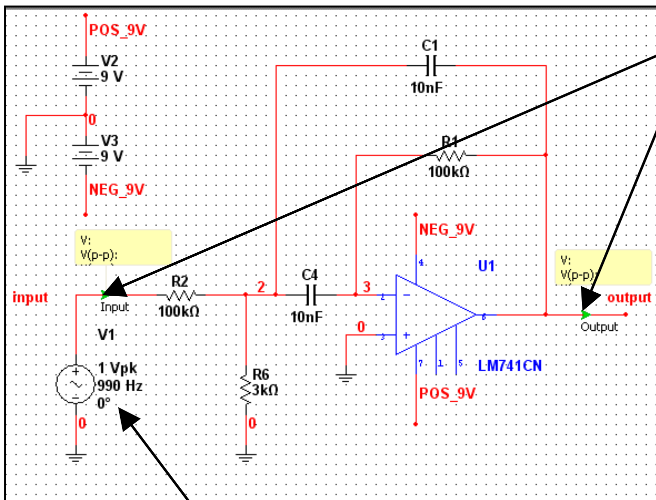
Summary: Most Standard or Advanced LabVIEW toolkits can be used in conjunction with Multisim for stimulus and measurement capabilities. (Follow templates given in Multisim)



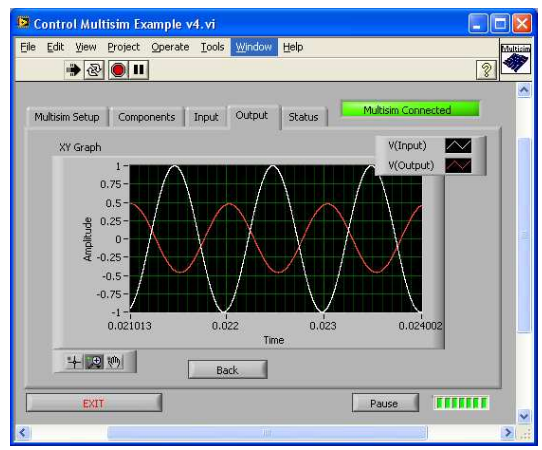
Advanced Control Design (,system ID, Control Design, dynamic system simulation, etc)		**Order Analysis** (Order Tracking, Spectrum Selection, Tachometer Processing, Waterfall, Orbit / Polar Plots, Bode Plots, etc)
Digital Filter Design (FIR / IIR Filter Design, Quantization, Fixed-point Modeling/Simulation, 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)

Multisim and LabVIEW For Virtual Device Prototyping

- **Why?** Valuable in Biomedical Research to start building validation plan and begin test development **BEFORE** actual prototype completion
- Multisim API and LabVIEW interface allows you to start test development **In Parallel** with design
- How it works: 1. **Build Circuit** Simulation in Multisim



3. Insert **Probes** in Multisim to be used as **Test Points**



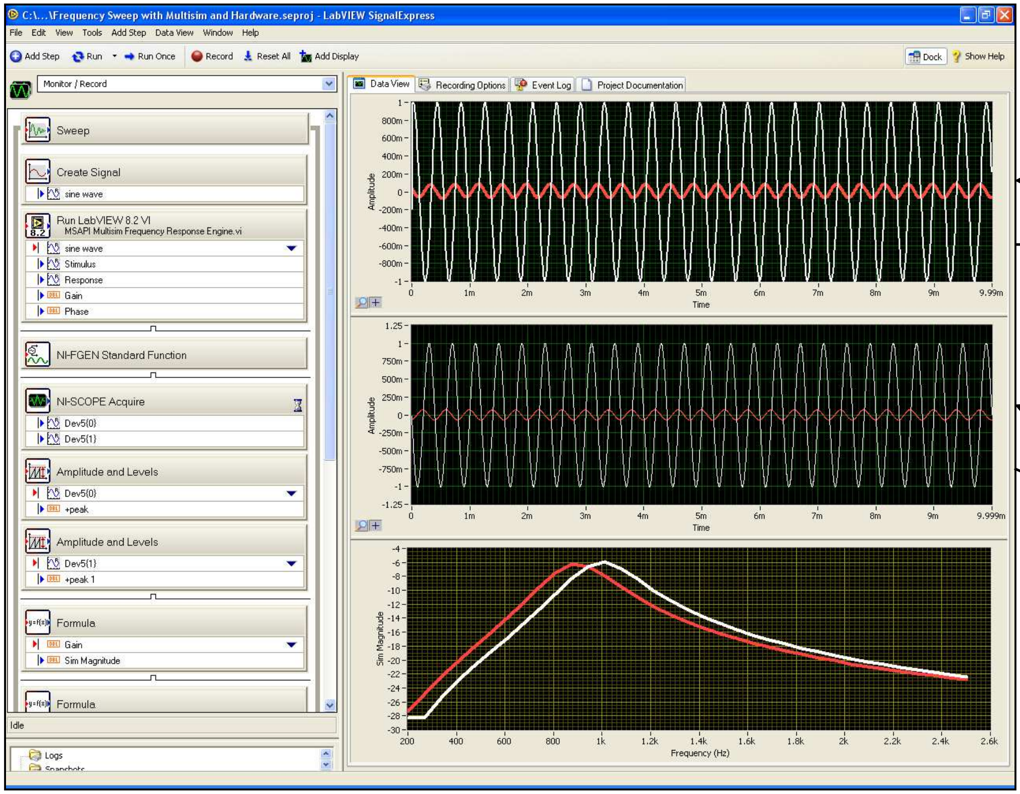
2. Insert appropriate **Test Sources** into Simulation

4. Use **LabVIEW VI** to Control Sources and Measure **Test Points** while simulating (similar to DAQ Sampling)

Virtual Device Prototyping

- **Direct Comparison** of Simulation Data *in-step* with Prototype Measurements
- Create a **Frequency Response** Profile with Hardware and Simulation

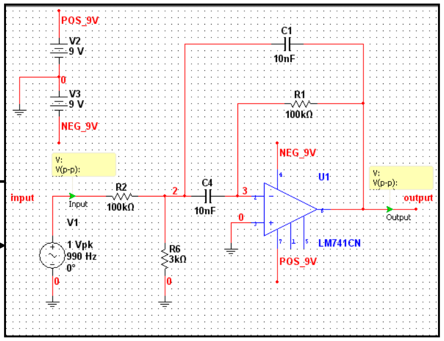
LabVIEW Signal Express Controlling Simulation and Measurement Hardware and Comparing Results



LVSE Script Controls Frequency Sweep for Multisim and Hardware

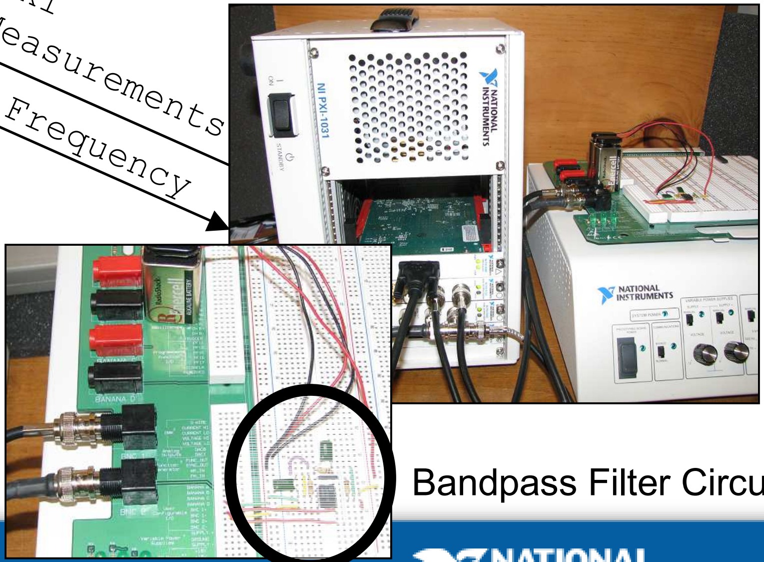
Multisim Simulation

Frequency



PXI Measurements

Frequency

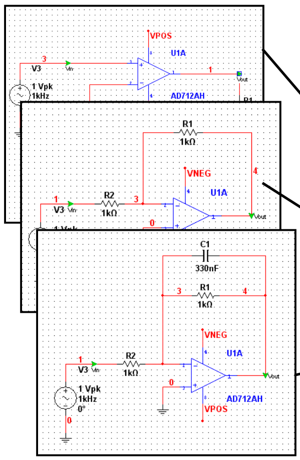


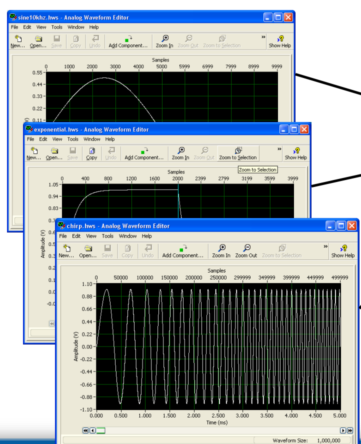
Bandpass Filter Circuit

Design Automation

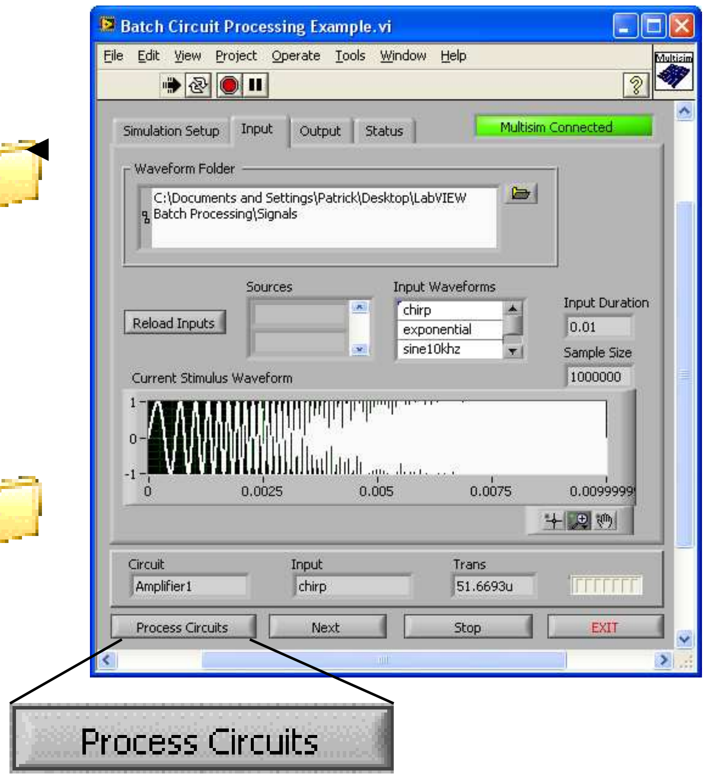
- Design Automation and Optimization using Multisim and LabVIEW

1. Create Circuits (Multisim) and Test Waveforms (Analog Waveform Editor)

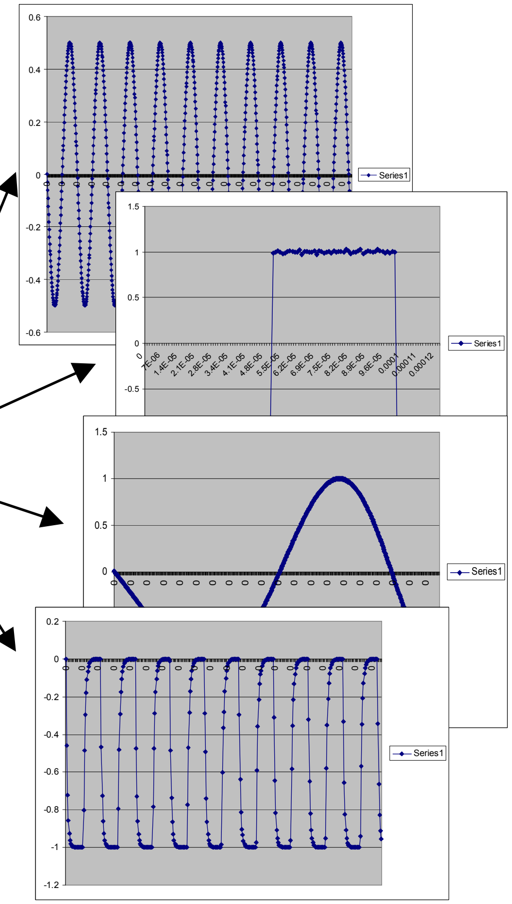




2. Automatically **Batch Process** Circuits and Stimulus Waveforms within Folders



3. Simulation Results Stored for Analysis



Conclusions

- Most Design Engineers use Simulation and Measurement Data *Separately* in the development of Circuits and Systems

- SPICE and Virtual Instrumentation can be **combined** to utilize the mathematical capabilities of **SPICE** and Measurement capabilities of **Virtual Instrumentation**

- Example Biomedical applications were shown using Multisim and LabVIEW demonstrating how to employ a unified simulation, validation and test strategy using SPICE and Virtual Instrumentation

For product information, go to ni.com/multisim