Closing the Hardware Design Loop with Electronics Workbench Multisim and National Instruments LabVIEW

By Shauna L. Rae, Electronics Workbench, and Gretchen Edelmon, National Instruments
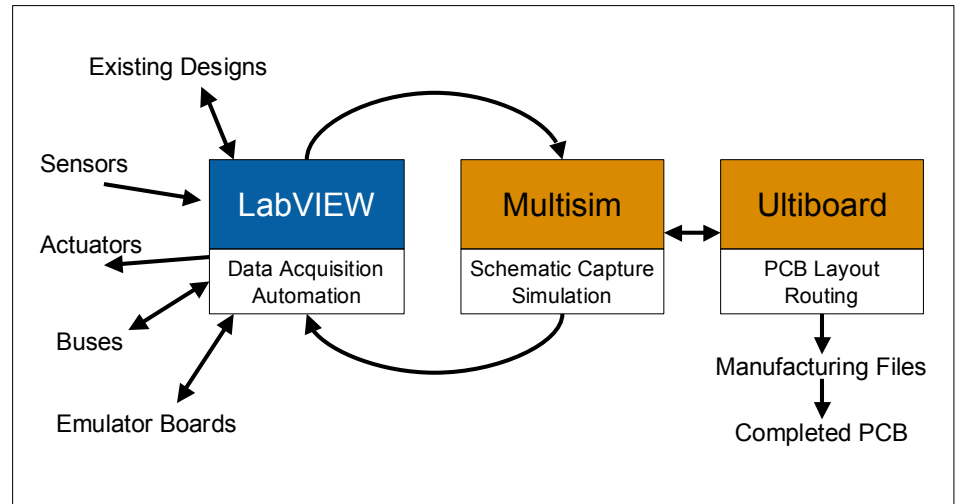
Designers are always looking for ways to improve their circuit simulations. The combination of Electronics Workbench Multisim and National Instruments LabVIEW allows engineers to close the hardware design loop. It provides designers with input and feedback never before available, allowing them to efficiently test and verify both their circuits and simulations.

Historically hardware design engineers have simulated circuits with models that are based on the electrical or physical characteristics of the components. These simulations have been very powerful and helpful in the design and verification process, but they are not without limitations.

As designers continue to look for ways to reduce time to market for new designs, the crossover point between the design and the characterization and validation process of actual hardware has been neglected. By bridging between the design and measurement area, users can iterate more tightly through the design process by identifying and addressing discrepancies and addressing them in the prototypes. The way to accomplish this is to be able to reuse and compare the signal data, simulated or measured, easily in either domain.

It has been difficult and time consuming to accurately model real world signals such as those from sensors or buses. It would be beneficial to capture these signals and use them to drive simulations. Simulation waveforms could in turn be used to drive existing interfaces to verify that the design is working according to specifications. It would also be useful to be able to compare simulation waveforms to real world waveforms in a direct format. This would help to tighten simulations and aid the debugging process.

This paper will provide a brief introduction to Electronics Workbench and their printed circuit board (PCB) design and simulation tools and to National Instruments with their measurement products, including modular instruments and Data Acquisition (DAQ) boards, programmed using their LabVIEW product. It will showcase how they work together and highlight some general applications.

Printed Circuit Board Design

Electronics Workbench develops Electronics Design Automation (EDA) tools for all major steps in the circuit design flow: schematic capture, simulation, PCB layout, autorouting, and CAM preparation.

Designers use schematic capture software to encapsulate their designs. Once a design is captured, they can use the schematic diagrams to drive simulations and/or feed the PCB layout and routing process.

Simulation programs are used to test and verify designs. Simulation helps designers to: reduce time to market, find errors in their designs before they become costly mistakes, evaluate problems potentially caused by variations in real world component properties, and optimize their designs.

Once a design is verified at the schematic level, it must be placed and routed for PCB production.
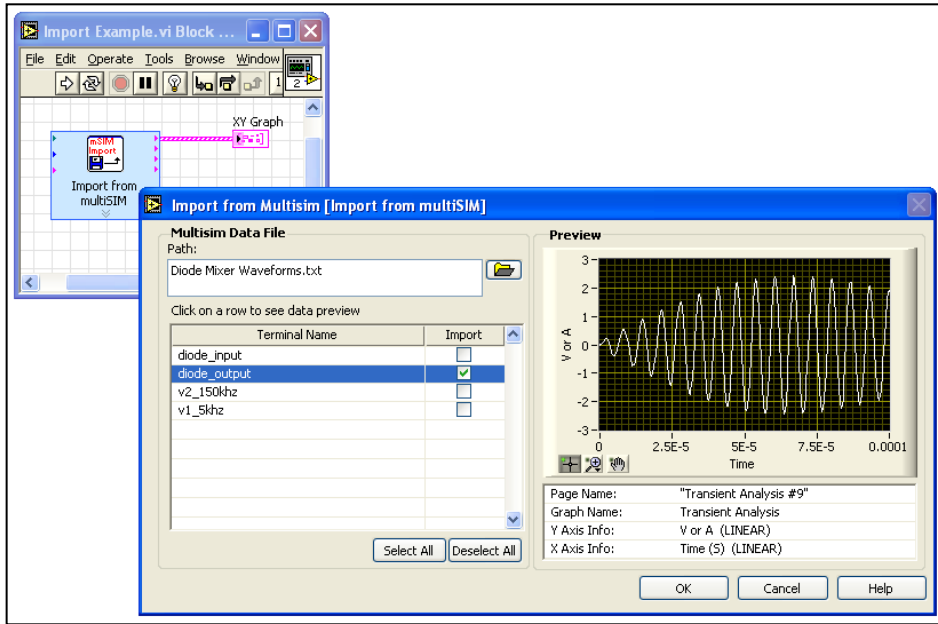
Multisim provides intuitive schematic capture, SPICE, Verilog, VHDL, and RF simulation, co-simulation, and analysis. With Ultiboard and Ultiroute, users perform automatic or manual placement and routing of components and traces at the PCB level. Ultiboard also outputs manufacturing files.

Simulation with Multisim is powerful yet easy-to-use. Virtual instruments connect to wires in the schematic diagram just like real world lab instruments connect to wires in a circuit. Virtual instruments in Multisim have front panels and controls just like their real world counterparts. If advanced users want more control, they still have access to traditional SPICE simulation and analysis methods.

Circuit simulators are powerful tools that use models to represent the behaviour of components. Alone, they do not have the capability to capture or drive real world signals. Designers need to make use of data acquisition and control devices and associated application software in order to do this. Until recently there was no easy way to do this.

Virtual Instrumentation with LabVIEW

LabVIEW is the leading software environment for data acquisition and instrument control. With LabVIEW,

Import Example.vi Block ...
File Edit Operate Tools Browse Window

XY Graph

mSIM Import

Import from multiSIM

Import from Multisim [Import from multiSIM]

Multisim Data File
Path:
Diode Mixer Waveforms.txt

Click on a row to see data preview

Terminal Name	Import
diode_input	☐
diode_output	☑
v2_150khz	☐
v1_5khz	☐

Select All   Deselect All

Preview

V or A

Time

Page Name:	"Transient Analysis #9"
Graph Name:	Transient Analysis
Y Axis Info:	V or A (LINEAR)
X Axis Info:	Time (S) (LINEAR)

OK   Cancel   Help

users can acquire, analyze, and present real-world data from plug-in data acquisition boards, modular PXI instruments, or traditional GPIB instruments. LabVIEW is commonly used to characterize circuit prototypes in design labs. Designers can easily create programs that generate analog and digital stimulus signals for their boards, and measure and analyze the response signals. These programs, called virtual instruments, combine powerful acquisition and analysis hardware and software on a standard PC. Many of these systems use plug-in data acquisition (DAQ) boards from National Instruments, which enable users to connect their circuits or sensors directly to their computer for an integrated measurement system.

LabVIEW is a complete programming language, so users have the flexibility to define exactly what they need – they are not limited by the predefined functionality of traditional GPIB instruments. LabVIEW users combine the measurement and analysis components they need through software. The key to enabling scientists and engineers to do this is through the graphical programming language of LabVIEW. LabVIEW is programmed using block diagrams, rather than a traditional text-based programming language which can take years to master. Engineers can quickly assemble programs by "wiring" together functional blocks, or VIs, for performing acquisition, analysis, and display functions in LabVIEW.

Integrating LabVIEW and Multisim

By combining the schematic capture, simulation, and analysis tools available in Multisim and the data acquisition and measurement features provided by National Instruments DAQ devices and LabVIEW, real world analog and digital signals are easily sampled and then incorporated into simulations. This means that designers can drive their simulations, including HDL testbenches, with data acquired from sensors, test points, or even data and address buses.

Designers can also import simulation results from Multisim into LabVIEW. There are two main reasons for doing this. The first is to compare simulated and real data and the second is to drive external circuitry with simulation results. Both can significantly increase the efficiency of the design verification process.

National Instruments now provides pre-built LabVIEW VIs that allow users to read in and export data from and to Multisim. VIs compatible with LabVIEW 6.1 are currently available and Express VIs are available for LabVIEW 7.

The VIs for LabVIEW 7 take advantage of a new feature called Express VIs. Express VIs are special VIs that allow users to easily configure complex operations through intuitive configuration panels. Specifically, for interfacing with Multisim, there are three Express VIs: 1) *Import from Multisim*, 2) *Resample Multisim Data*, and 3) *Export to Multisim*.

The VIs serve one main purpose. Multisim and LabVIEW have different default data formats. Because of the nature of SPICE simulation, Multisim simulation data is not evenly spaced; it is represented in the form of XY data pairs. LabVIEW data is uniformly distributed and each data set is represented first by the delta value between two consecutive data points and then the data points.

The *Import from Multisim VI* reads in Multisim format simulation results into LabVIEW without converting them to the LabVIEW data format. In many applications this is not a hindrance and the XY pair data can be plotted directly onto a graph. If the user wants to perform advanced operations or analysis in LabVIEW, they would need to use the *Resample Multisim Data VI*. This VI converts the XY data pairs into evenly distributed LabVIEW format data.

The *Export to Multisim VI* converts the data from LabVIEW format to the Multisim required XY data format and saves it to a file. The user can then use the Piecewise Linear Source in Multisim to read in LabVIEW data.

Possible Applications

- Designs with sensors
- Designs that interface with existing circuitry
- Designs with bus interfaces
- HDL designs
- Design reuse
- Debugging

Applications

Sampling Signals

As mentioned before, National Instruments excels at capturing real world signals and bringing them into a computer for further processing. This is helpful when you want to capture real world output from sensors. It is the only way that you can drive your circuit simulations with sensor values that would actually occur in the real world.

Many designs are not first generation designs, but rather modifications or extensions of existing ones. There are also a vast number of designs that include modules from past designs. Using LabVIEW and Multisim designers can accurately represent existing circuitry in their simulations. This is extremely useful when portions of the designs cannot be easily modelled for simulation. The same applies to prototypes of circuits and to signals from

goes both ways. This means that not only can users sample values but they can also bring their simulation results over to LabVIEW. This provides an opportunity to easily compare simulation results to real world waveforms. Some of the benefits include increased debugging efficiency, design verification, and improving simulations.

Designers can cross reference the actual performance of a system to the desired performance. This can help them track down design or manufacturing errors faster. It can also help designers understand which simulation models are too idealized and adjust them accordingly.

The following example illustrates this concept. In this example, the actual performance of an operational amplifier phase shift oscillator is compared to the behaviour predicted by a simulation of the circuit. As shown in the schematic diagram, this is a common operational amplifier circuit with three RC networks in its feedback path. Each of these RC networks introduces a 60° phase shift thus producing the 180° phase shift required for oscillation.

Two different aspects of circuit behavior were simulated and then compared to actual data. Both of these comparisons were made by first running a transient analysis in Multisim to see the time domain response of the oscillator and then importing the simulated data into a LabVIEW VI programmed to acquire the data from the protoboard circuit.

The first comparison made in this example is the oscillation frequency. The circuit should oscillate at a specific frequency based upon the RC network values. The LabVIEW VI front panel displays both the simulated frequency value and the actual frequency measurement and percentage of error between the two values.

Secondly, the example plots the simulated and measured waveforms from four important nodes in the circuit: the operational amplifier output and the output of each of the three RC networks. By plotting these waveforms along side
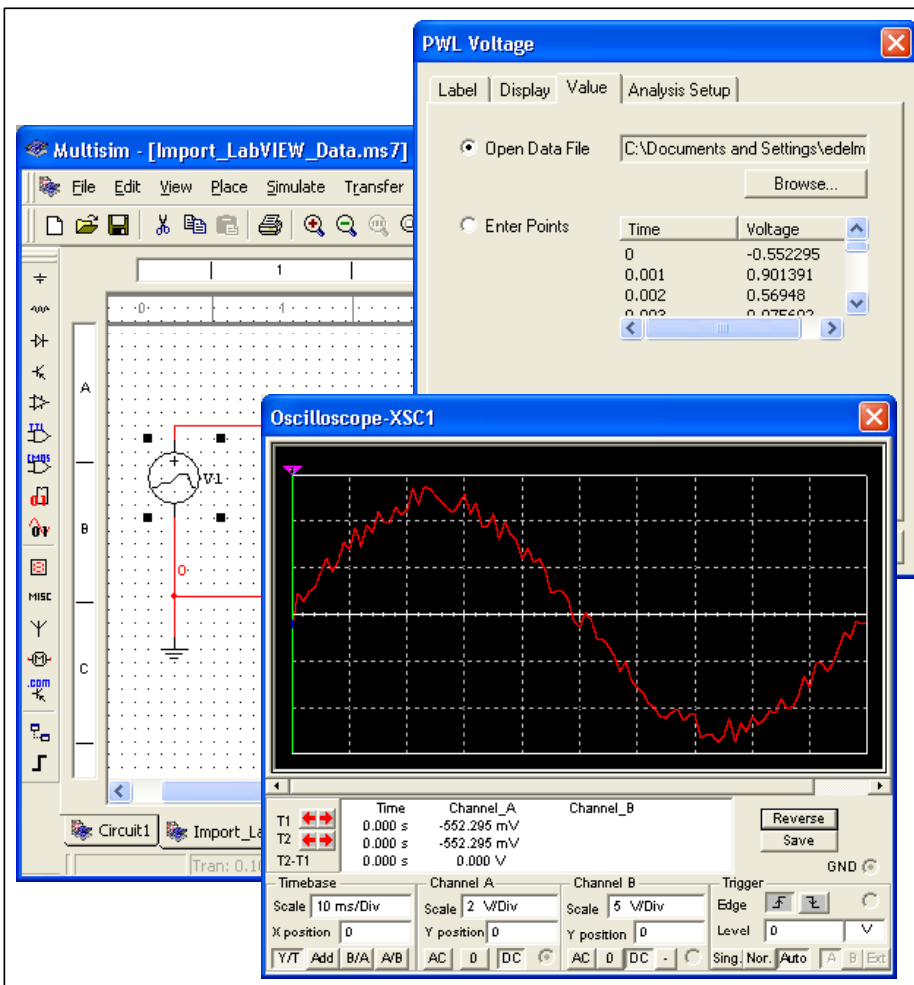
The integration between I/O Devices and circuit simulation applications also enables users to accurately develop bus or other digital signal interfaces. Designers can sample actual read/write cycles from an existing bus. This allows them to understand which lines of the bus are relevant to the interface and extract the critical timing information with a great deal more ease than looking up and dissecting timing diagrams.

microprocessor or microcontroller emulator boards.

Because Multisim allows users to co-simulate HDLs and SPICE, designers can take advantage of this relationship to create real world HDL testbenches.

Comparing Simulation and Real World Values

The relationship between the two applications, LabVIEW and Multisim,

each other, the designer can visually verify if the signals match to within allowable tolerances. Additionally, the example specifically calculates the phase shift between any two nodes that the user selects. The simulated difference is displayed along with the measured difference and a percentage error.

Once designers have a both the simulated and measured data together in LabVIEW, they can begin an iterative process to refine and perfect their designs. If the circuit behavior does not match the specification, they can return to the Multisim design environment. There they can adjust component values or models and perform additional simulations in an effort to match the simulation results to the real world data. Additionally, they can measure the actual stimulus signals at various nodes on the circuit and export that data for use in Multisim. This step will allow them to perform a new simulation with the actual data acquired from the circuit as the signal source instead of an ideal input source.
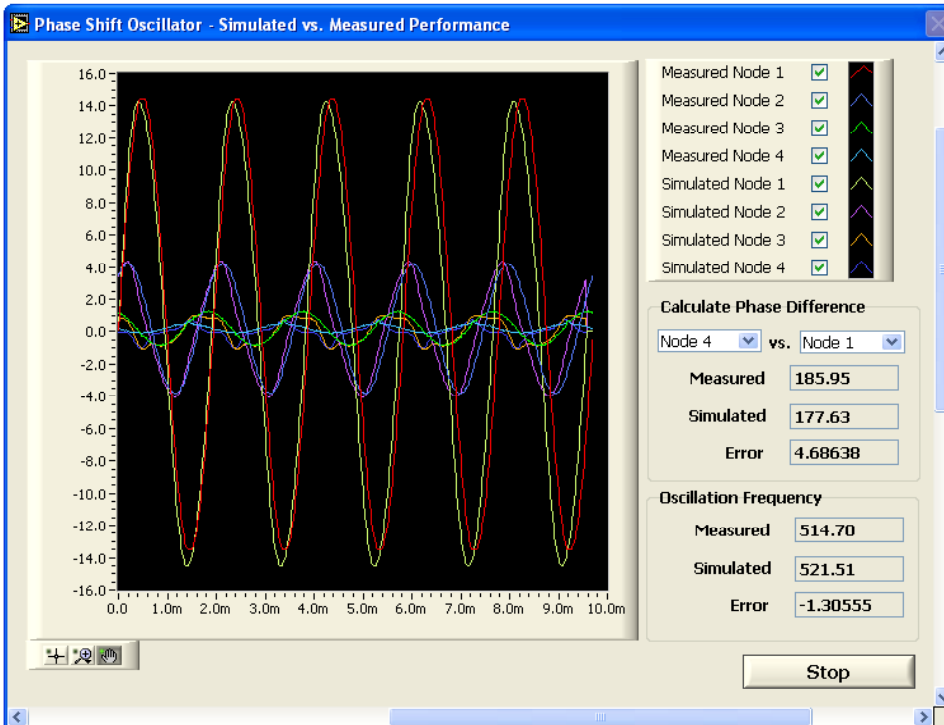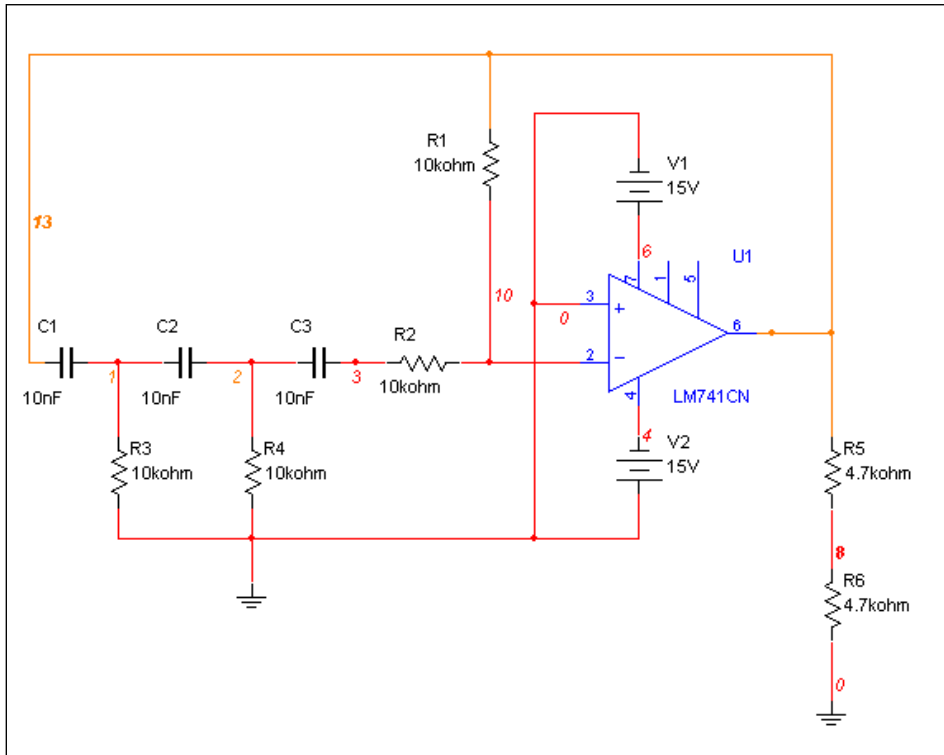
Summary

By taking advantage of both LabVIEW and Multisim, designers can improve and quicken the circuit design process. Using both programs allows users to perform:
- Data acquisition using National Instruments DAQ boards and LabVIEW
- Simulation with Electronics Workbench Multisim
- Driving simulations with sampled analog signals (Analog buses)
- Driving simulations with digital signals including data and address buses
- Driving simulations with microprocessor signals from an emulator board
- Driving HDL testbenches with real world values
- Comparing simulation waveforms to real world values
- Tightening the accuracy of the simulations
- Debugging by comparing simulation waveforms to actual ones
- Driving external circuitry with simulation results

Contact Information:

Electronics Workbench:
www.electronicsworkbench.com
Sales: 800.263.5552 or
nasales@electronicsworkbench.com
Technical Support: 416.977.5550 or
support@electronicsworkbench.com

National Instruments:
www.ni.com
Sales: 800.433.3488 or info@ni.com
Technical Support: 800.433.3488 or
http://www.ni.com/ask