Technical Article
MS-2207

A Recipe for Better System Block Design—Add SPICE

by Reza Moghimi, Applications Engineering Manager, Analog Devices, Inc., and Natasha Baker, Product Marketing Engineer for Multisim, National Instruments

IDEA IN BRIEF

Simulation has become an essential stage in the design process because it lets engineers evaluate and validate circuit behavior before prototyping, prevents design flaws from cascading through the design chain, and helps designers improve the performance of their circuits in a virtual environment, risk-free.

Is there anything more frustrating than having the board shop kick back an error-ridden design? Many designers today are under pressure to produce a prototype in weeks (if not days), and there's a limited margin for design iterations. Fortunately, the latest design tools improve productivity by providing a holistic and intuitive approach to circuit design and validation.

Many semiconductor manufacturers provide tools to aid in the design of robust system blocks during the initial specification stage. Analog Devices,Inc. (ADI), for example, has an online filter design tool (see Reference 1) that guides users through the process of active filter synthesis and the selection of recommended op amps based on those specifications. The tool then generates the final design topology, along with a bill of materials and SPICE netlist. In the stages before prototyping, simulation environments such as those from National Instruments (NI) provide further optimization and validation using macromodels of the specified parts (see Reference 2).

In this article, we explore how this holistic approach can speed up and improve the often daunting task of filter design—a common building block in a range of electronics applications. But first, some background.

SIM BASICS

The most popular analog circuit simulation tool is SPICE, which stands for simulation program with integrated circuit emphasis. SPICE dates back to the late 1960s when it was developed at the University of California, Berkeley. SPICE evolved into the industry standard for analog circuit simulation and remains the world's most widely used circuit simulator. Over the years, more simulation algorithms, component models, and extensions have been added. XSPICE, for example, developed at Georgia Tech, allows the behavioral modeling of components to accelerate the speed of mixed-mode and digital simulation. The NI Multisim™ environment supports both SPICE 3F5 and XSPICE simulation.

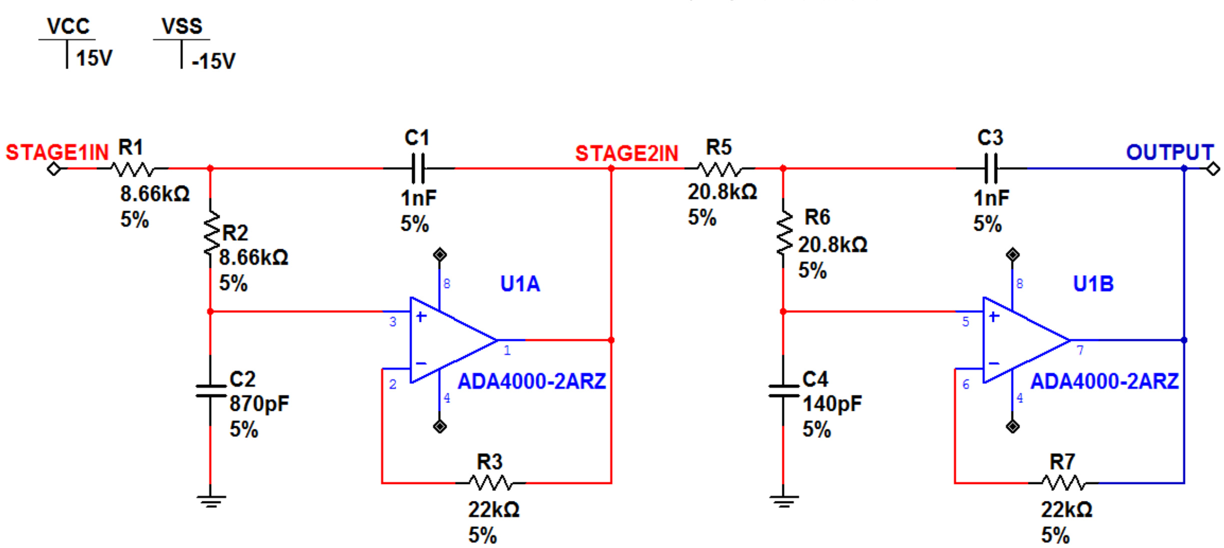

Figure 1. 20 kHz Butterworth Filter in NI Multisim

But why should designers bother with simulation? Simulation has become entrenched as an essential stage in the design process because it lets engineers evaluate and validate circuit behavior before prototyping. Simulation prevents design flaws from cascading through the design chain to fabricated boards where redesign becomes exponentially more expensive. Furthermore, through the exploration of a range of what-if scenarios, designers can improve the performance of their circuits in a virtual environment, risk-free.

One of the main benefits of using a circuit simulator is the ability to simulate macromodels that emulate real, orderable parts. Modern SPICE simulators are also taking an increasingly graphical approach to what has traditionally been a text-based process. For instance, NI Multisim incorporates more than 17,500 components, with many macromodels from leading semiconductor manufacturers; a text-based SPICE netlist is automatically generated upon capturing a circuit, and interactive measurement instruments, such as the oscilloscope or function generator, have displays and functionality that mimic their real bench-top counterparts. With these graphical extensions, designers no longer need to have expertise in SPICE syntax to leverage the benefits of simulation.

SIMULATION AND FILTER DESIGN

Filters are everywhere—from ultrasound equipment to pacemakers, where it is vital that only a specific range of frequencies be passed. Yet, while filters are a ubiquitous building block of electronics applications, filter design is little understood and often painful. What makes it so complex? Often, the required filter order for a specific performance is not well understood by system designers whose strength is not analog circuit design.

There are many variations in filter types (for example, Butterworth, Chebyshev, and elliptic) that are optimized for various specifications such as monotonic ripple or transition region width. Filter design also involves writing complicated math equations for identifying pole/zero locations that change the filter shape (see Reference 3). Another wrinkle is that the perfect components assumed during theoretical calculations do not exist; for example, manufacturing tolerances of resistors can affect expected circuit behavior.

Design tools such as filter wizards greatly simplify this complex task by helping designers understand the differences between the different topologies, as well as suggest parts for use in the design, without requiring complicated math. Graphical environments let designers observe how their circuit will operate across a wide range of component tolerances.

VALIDATING THE DESIGN OF A BUTTERWORTH FILTER

In our example, we validate the design of an active filter. The filter was designed using the ADI Filter Wizard and incorporates the ADA4000-2 dual precision op amp, which has been selected for its fast slew rate and stability with capacitive loads, making it a perfect fit for filter design. This op amp's pico-amp bias current allows the usage of high value resistors to construct low frequency filters without needing to worry about adding to dc errors. Additionally, the high value used for R1 minimizes interaction with signal source resistance. Higher order filters are possible by cascading more blocks; however, sensitivities to component values and the effects of interactions among the components on the frequency response increase dramatically, making these choices less attractive. The signal phase is maintained through the filter (noninverting configuration).

The filter was captured in NI Multisim for validation and further analysis (see Figure 1). This low pass, fourth order, Butterworth filter was designed with a 20 kHz cutoff frequency and Sallen-Key implementation due to its ease of design, maximally flat frequency response, and minimal component requirements. Butterworth filters are monotonic in pass band and stop band and have optimal pass-band ripple and a wide transition region (that is, the region between the pass band and stop band). They are frequently used as antialiasing filters in data acquisition systems. A two-pole version of Sallen-Key filter topology is used on the EVAL-FLTR-SO-1RZ and EVAL-FLTR-LD-1RZ filter boards, which can be ordered from ADI. The application note for this board is AN-0991.

When designing a filter, it is important to consider both the frequency and time-domain responses of the circuit. Let's investigate how we can validate these characteristics using NI Multisim.

Validating Frequency Response

Figure 2 illustrates the results of an ac analysis. The simulation results indicate a cutoff frequency (the frequency at which the gain drops by 3 dB) of 20.1 kHz, which closely approximates the 20 kHz specification we set out for. We can see that beyond this corner frequency, the gain falls off at 80 dB per decade (−20dB/dec or −6dB/oct for each pole in the filter's transfer function).

We also observe that the stop band does not continuously decrease as we would expect from an ideal filter; the gain begins to increase at approximately 1 MHz due to the loss in op amp voltage gain. Using the cursors, we estimate this stop band to be approximately 700 kHz.

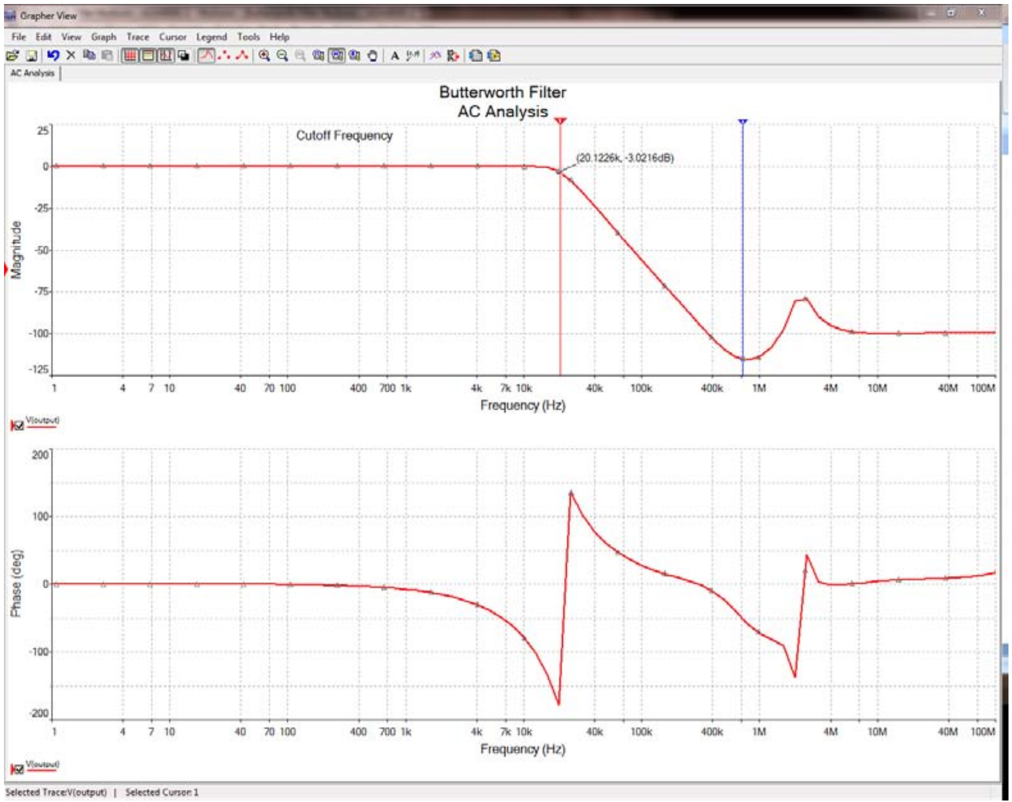

Figure 2. Frequency Response of the Butterworth Filter

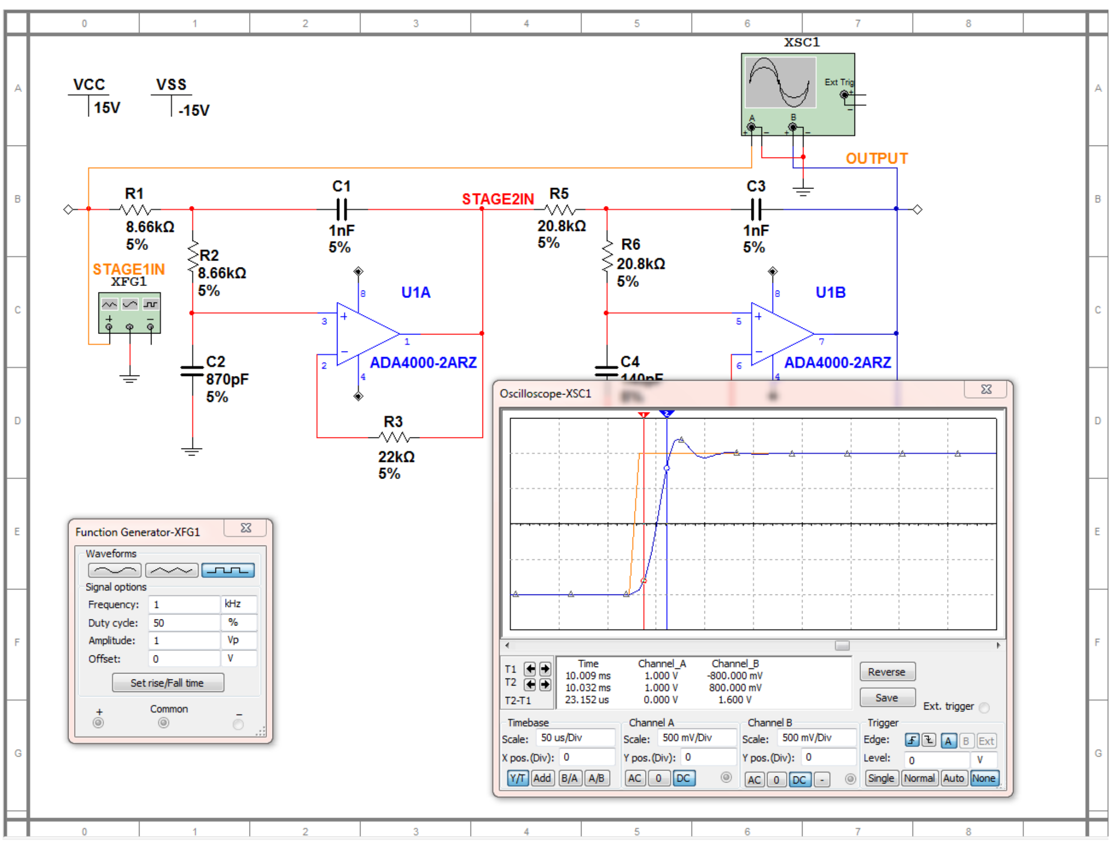

Figure 3. Investigating the Time-Domain Response Using Virtual Instruments

Validating Time-Domain Response

We can investigate the step response using measurement instruments available in Multisim. The function generator allows us to input a stimulus, and the oscilloscope allows us to observe our output waveform, both directly within the schematic environment. These measurement instruments mimic their bench-top counterparts; with the oscilloscope, for example, parameters such as the time base and voltage divisions can be adjusted based on waveform characteristics. With measurement instruments, we can also change settings in real time, such as the frequency set by the function generator, which allows us to see how much our signal is being attenuated for a frequency beyond the 20 kHz point.

We can measure characteristics such as rise time and settling time with the oscilloscope, as shown in Figure 3; however, we can also view this data within the Grapher, an option that allows us to annotate and print the graphs for documentation purposes.

The first characteristic we investigate is rise time (defined as the time from 10% to 90% of its final output value); using the cursors, we determine this to be 19.3 µs. We also see a settling time of approximately 92 µs. These characteristics are annotated on the graph shown in Figure 4. (Note that Parameter TMAX affects rise time, and was changed from the default for the purpose of this example.)

Taking Worst Case Scenarios into Account

Another core benefit of simulation is the ability to account for nonideal component values (that is, tolerances). In this section, we execute a Monte Carlo analysis that runs multiple ac analyses using permutations of component values within the 5% component tolerance range we defined in our schematic; this allows us to see how our cutoff frequency is affected in the worst cases. (Note that this analysis can also be run for a transient or dc operating point analysis).

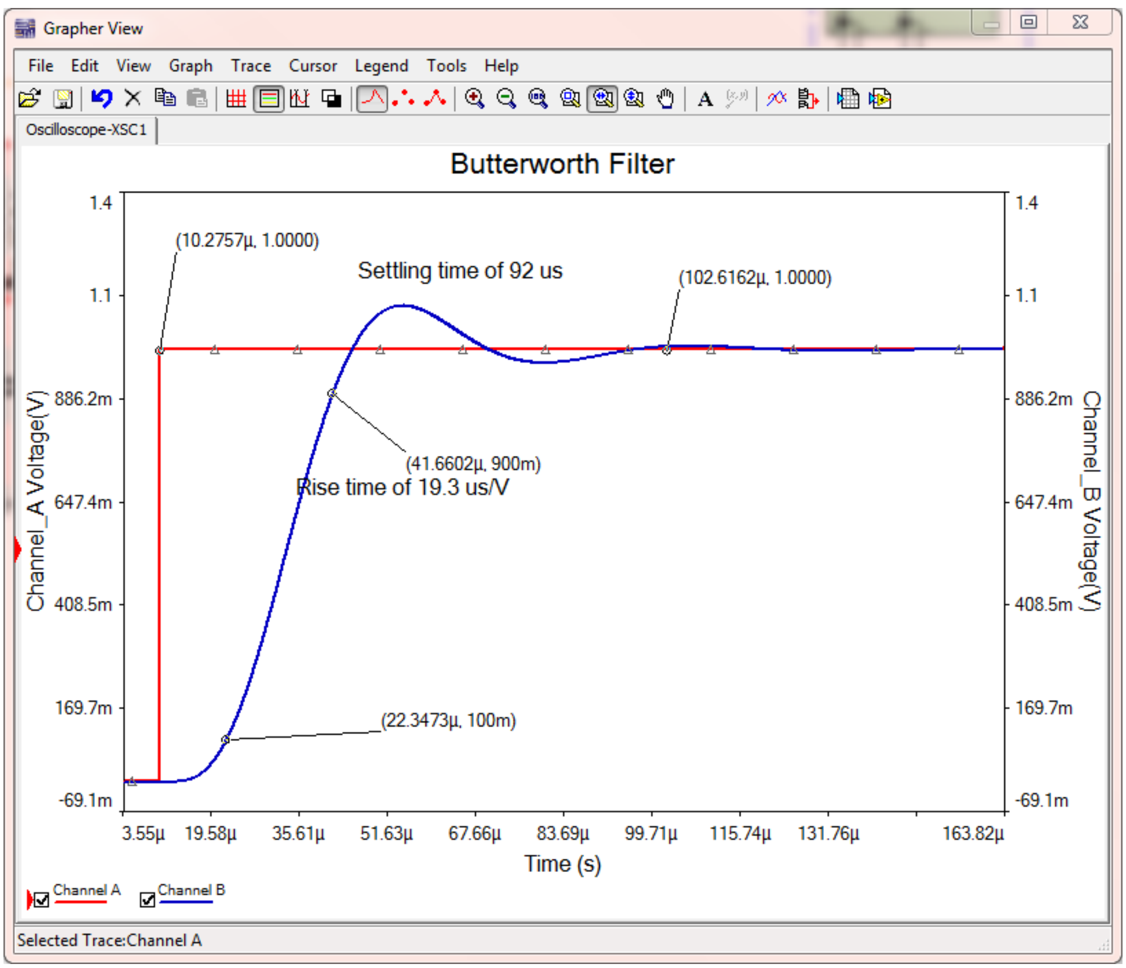

Figure 4. Documenting Time-Domain Characteristics Using the Grapher

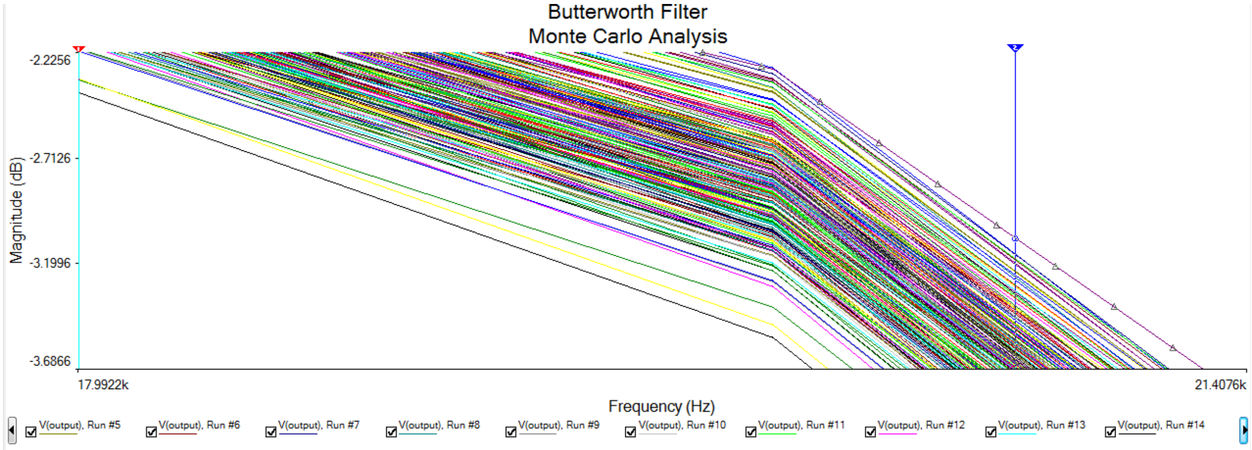

Figure 5. Monte Carlo Simulation Results

The first run is a nominal run, assuming ideal conditions. The output of our analysis, which iterates upon 200 permutations of our circuit, is shown in Figure 5. Observe that the 171st run (bottom trace) and 2nd run (bottom trace) define our worst cases with cutoff frequencies of 20.67 kHz and 19.02 kHz, respectively. This deviation in cutoff frequency presents a low sensitivity of this filter design to component variances.

As we have seen, some measurements require more post-processing than others. Tasks such as calculating rise time, for example, can become tedious if done repetitively. Fortunately, there are tools that solve that problem as well. NI LabVIEW™ is a graphical programming language that allows us to create a custom interface for visualizing and analyzing measurements within Multisim. This instrument automates the calculation of rise time, slope, overshoot, and undershoot of a filter design based on its input and output waveforms. By creating a custom instrument, designers can automatically display accurate values of characteristics that would traditionally require manual postprocessing. Custom instruments can be made for a broad range of applications, including the import of real acquired measurements into NI Multisim that incorporate real-world effects, such as noise, for even greater simulation accuracy.

CONCLUSION

Today's system designers cannot afford to run with unverified ideas. With modern design tools, such as the ADI Filter Wizard, already built and verified circuits (*Circuits from the Lab Reference Circuits* (CFTL available at http://www.analog.com/circuitsfromthelab), and NI Multisim, they don't have to. Engineers can validate and improve circuit behavior long before the prototyping stage, vastly improving design productivity. The result is fewer costly redesigns, faster time to market, and better design performance.

REFERENCES

Reference 1

Analog Filter Wizard™ Design and Products Selection Tool, V1.0 available at http://www.analog.com/en/amplifiers-and-comparators/products/dt-adisim-design-sim-tool/Filter_Wizard/resources/fca.html. ADI also provides active filter evaluation boards to enable the quick prototyping of two-, four-, or six-pole low-pass or high-pass filters, with frequencies ranging from kilohertz to tens of megahertz.

Reference 2

NI Multisim Component Evaluator – Analog Devices Edition is a free version of the circuit design and simulation environment for the purpose of evaluating ADI components Available at http://www.analog.com/nimultisimevaluator.

Reference 3

Linear Circuit Design Handbook available at http://www.analog.com/library/analogDialogue/archives/43-09/linear_circuit_design_handbook.html.

One Technology Way • P.O. Box 9106 • Norwood, MA 02062-9106, U.S.A.
Tel: 781.329.4700 • Fax: 781.461.3113 • www.analog.com

T09990-0-7/11(0)