How to Select the Best Amplifier Using LTspice Simulation and Noise Analysis

Hooman Hashemi, Product Applications Engineer

Abstract

This article explores one of the simulation methods that can be utilized to identify and optimize device/component selection in noise sensitive applications. We will first point out one of the features of LTspice® simulation that allows this for standard components and then introduce an approach that allows this same capability for operational amplifiers (op amps) used in a signal path. Since low noise is often associated with higher power consumption and higher cost, this tool allows one to choose the lowest power and most cost-effective solution that meets the requirements of the design.

One of the challenges of designing a signal path that involves a transducer (small signal), amplifiers, filtering, and the data acquisition (ADC) is to determine the noise contribution of the various components and blocks that the signal traverses before getting digitized or otherwise processed. If the design is done correctly and most of the gain is taken at the front end, the task is simpler since choosing the lowest noise front end ensures the highest SNR and minimizes the impact of the rest of the circuitry.

But what if such a clear distinction cannot be made and/or the application necessitates the lowest levels of noise and signal integrity where noise must be fully optimized?

Noise Analysis of Standard Components

Engineers use LTspice effectively already for noise impact study and optimization since LTspice allows you to click on a particular device (that is, resistors and transistors) in a simulation and immediately see its output noise contribution as a noise density plot in noise analysis.

Figure 1 shows an example simulation where the output noise and other resistor noise contributions are plotted side-by-side to easily see the relative importance of each compared to the net output noise. Additionally, it is possible to integrate any of those noise density plots to see its impact over the entire frequency range, as shown for R3 as the highest resistive contributor at 100 nV/$\sqrt{Hz}$ flatband.

LTspice is a very powerful simulation tool covering resistors and transistors, as far as modeling and keeping track of their noise contribution. However, you may need an alternative solution for considerations for other components/building blocks. That would be the case for devices such as op amps with encrypted macromodels that may be scattered throughout your circuit and in your simulation file.

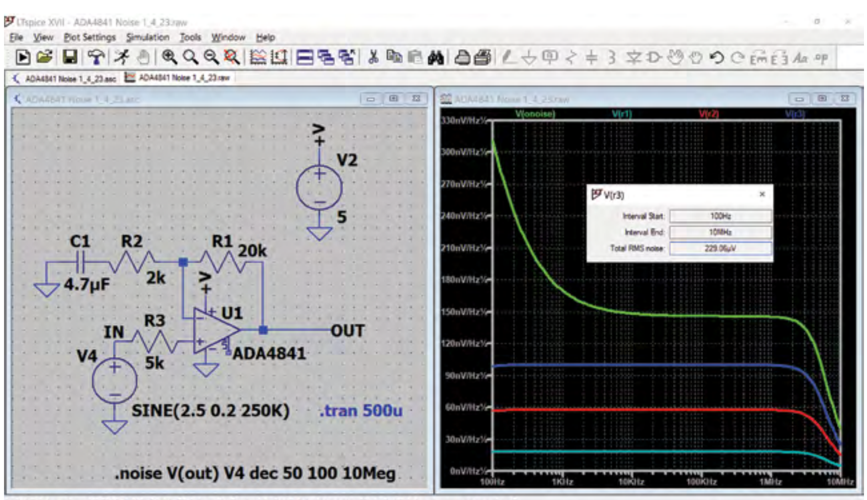

Figure 1. Simulated output noise along with each resistor's noise contribution.

To fully illustrate what is meant here and how to do noise analysis for an op amp, it's necessary to first introduce how one can add an idealized op amp in LTspice. Figure 2 shows the UniversalOpAmp.asc file built into the LTspice educational library. It shows a simplified op amp model with five levels of increasing complexity encompassing the simplest to the most complex version all outputs plotted simultaneously. This is a useful macromodel that can be copied into any design and can be easily manipulated/edited to find the impact of each parameter.

Extending the Analysis to Op Amps

The UniversalOpAmp can be a powerful tool in simulation when it comes to determining the expected noise impact of the device. By using this LTspice component, one can easily vary its voltage noise, current noise, and the respective corner frequencies of each noise source to see the resulting output/input noise. Armed with this information, one can go about intelligently picking the right device for the job by knowing the exact noise tolerance of the design.

Figure 3 shows this approach where the circuit of Figure 1 is modified to take the UniversalOpAmp instead. With the noise current "In" set to be a variable parameter, and the voltage noise term "En" value of 0.1 nV/√Hz to be insignificant, with the ".step param" function, one can see the result of varied current noise easily after simulation. In this case, the parameter is varied through a list: 0.1 pA/√Hz , 1 pA/√Hz , 2 pA/√Hz , 5 pA/√Hz , and 10 pA/√Hz.

Please keep in mind that this technique is a first-order approximation of an op amp's noise characteristics. Behavior such as rising noise with frequency, often found in FET input op amps, is not included in the universal model and must be accounted for separately once the actual device is simulated or bench tested. For more information on the FET noise current behavior, please refer to "Current Noise in FET Input Amps."

It is worth noting that in many applications, such as transimpedance amplifiers (TIAs) where noise performance is usually very important, there is a strong interaction between the amplifier noise and the external components (for example, input capacitance from a photodiode or avalanche photodiode (APD)) such that any such evaluation is accurate only if these external components are included and accounted for. Otherwise, the simulated performance will be far from the measured results!

One can then put together the following Table 1 simulation results summary and compare each case against the most dominant thermal noise (R3 at 100 nV/√Hz) as added noise by the amplifier. Here is the sample added noise computation of 0.1 pA/√Hz (Case 1) for reference:

$$Added\ Noise = 20 \times log(117\ \text{nV}/\sqrt{\text{Hz}}/100\ \text{nV}/\sqrt{\text{Hz}}) = 1.4\ \text{dB} \quad (1)$$

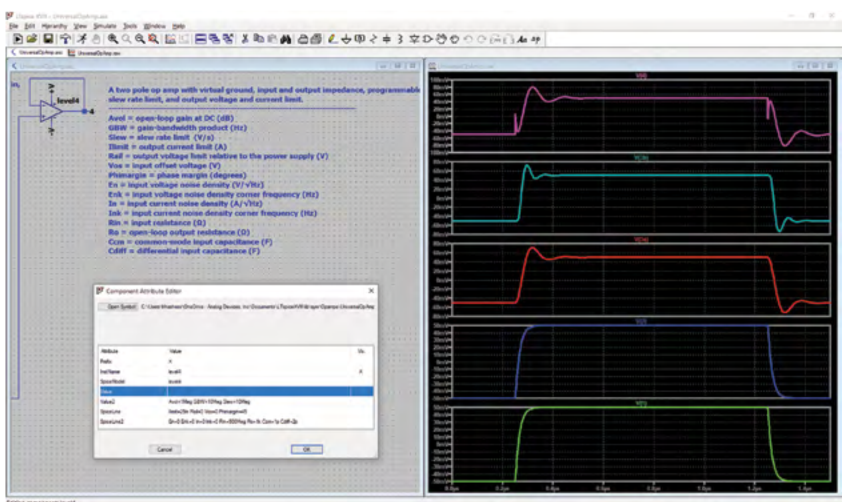

Figure 2. Universal op amp versions.

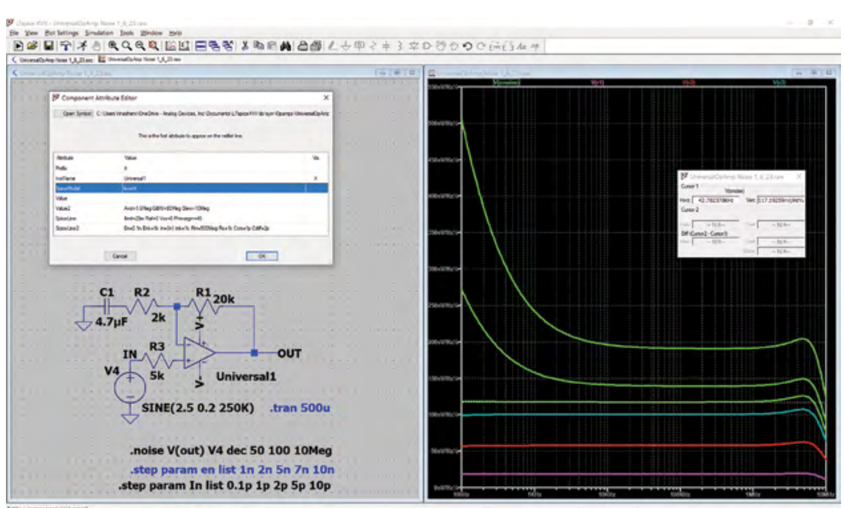

Figure 3. UniversalOpAmp with stepped noise current (first three values plotted).

Table 1. Noise Current Variation Summary

Case Number	Input Current (pA/√Hz)	Output Noise (nV/√Hz)	Added Noise (dB)
1	0.1	117	1.4
2	1	140	2.9
3	2	192	5.7
4	5	398	12.0
5	10	771	17.7

A similar kind of simulation can be done with voltage noise (while setting current noise low to be insignificant) varied and similarly tabulated, as shown in Table 2:

Table 2. Noise Voltage Variation Summary

Case Number	Input Voltage (nV/√Hz)	Output Noise (nV/√Hz)	Added Noise (dB)
5	1	117	1.4
6	2	119	1.5
7	5	130	2.3
8	7	141	3
9	10	162	4.2

Looking at Table 1 and Table 2, it is possible to conclude that keeping the added noise to less than or equal to 3 dB, for example, would require input current noise to be <1 pA/√Hz and input voltage noise to be < 7 nV/√Hz. A device such as the AD8055 with 1 pA/√Hz and 6 nV/√Hz noise fits the bill and, when simulated, has a broadband noise of 144 nV/√Hz that is close to the predicted noise from Table 1 and Table 2.

Summary

UniversalOpAmp, a tool/feature built into LTspice, was explored. This tool allows the circuit designer to vary the voltage and current noise of the prospective active amplifier used in an application to predict the resulting added noise. Armed with this information, when compared to other dominant noise sources in the design, one would then be able to pick the most suitable amplifier to hit the intended noise requirements at optimum power consumption and cost.

Reference

Win, Kaung. "Current Noise in FET Input Amps." *Analog Dialogue*, Vol. 54, No. 1, February 2020.

About the Author

Hooman Hashemi joined Analog Devices in March 2018, where he works on characterizing new products and developing applications that showcase the products' features and uses. Hooman previously worked for Texas Instruments for 22 years as an applications engineer, concentrating on the high speed portfolio. He graduated from University of Santa Clara with an M.S.E.E. in August 1989 and San Jose State University with a B.S.E.E. in December 1983.

TA24522-6/23