Dear Editor,

Recently, in Electronics World 2015-02, I've read with great interest Mike Engelhardt's article about LTspice's superiority to pSpice. A few weeks later, in LT's Journal of Analog Innovation, the same article appeared again, with a few extra remarks (Mr Engelhardt is LT's manager of simulation development). In addition, Bob Cordell's extensive and excellent Vol. 9 book review about Gilles Brocard's LTspice IV software book might even convince people like me who live in the habit of mistrusting all kinds of simulation software when it comes to electronic noise.

Thus, I thought it would be a good idea to check LTspice's ability to simulate the electronic noise behaviour of amps; of course, it did that by comparing the results with the ones of the math I know and understand and not by comparing it with pSpice results, because I don't use it for noise simulations.

LTspice contains a special noise analysis command ".noise ….", allowing one to calculate the output noise voltage density “V(onoise)” of an amp in a pre-defined frequency range. Division by "gain" should lead to the input referred noise voltage density “V(inoise)”, and application of another special command will lead to the rms value of the output noise in the chosen bandwidth (by integration). Hence, the calculation of the output referred signal-to-noise ratio will then become very easy.

I tried these approaches with simple op-amp circuits and they work fine with circuits like the one of **Fig. 1a**, an op-amp in series (inverting) configuration.

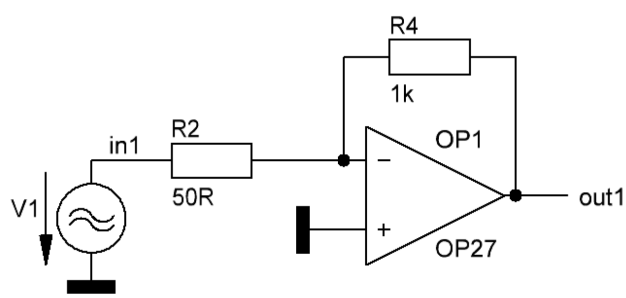

Fig. 1a Test circuit of an OP27 driven amp in inverting configuration

“V(onoise)” divided by the command "gain" leads to the correct input referred “V(inoise)”, because the gain is correctly set to the noise gain G_N = R4/R2+1 = 21, and not to the amplifier's signal gain R4/R2 = 20 = V(out1)/V(in1)!

Surprisingly, a BJT in a (signal inverting) common emitter configuration *à la* **Fig. 1b** shows the same "gain" application: to get V(inoise) of such a gain stage division by the "signal gain plus 1" is chosen by the software, which, in my eyes, is wrong in this kind of configuration; it should just be the signal gain.

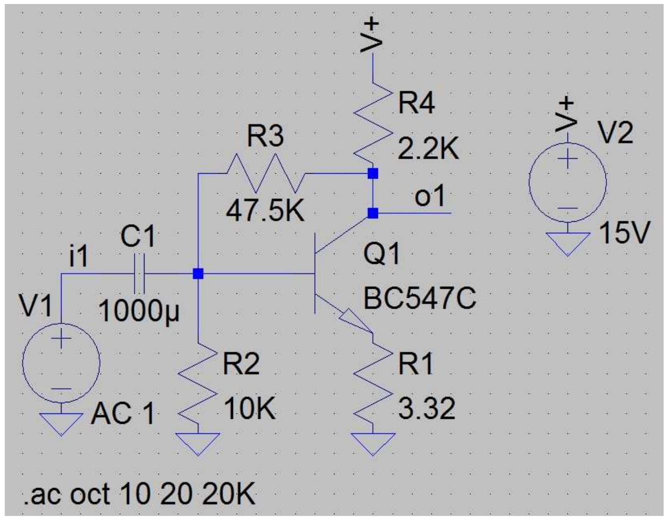

Fig. 1b Example common emitter gain stage

The next example in **Fig. 2** shows another picture. Here, the LTspice approach does not work right either. The op-amp works in a non-inverting (shunt) configuration, also with a signal gain of 20.

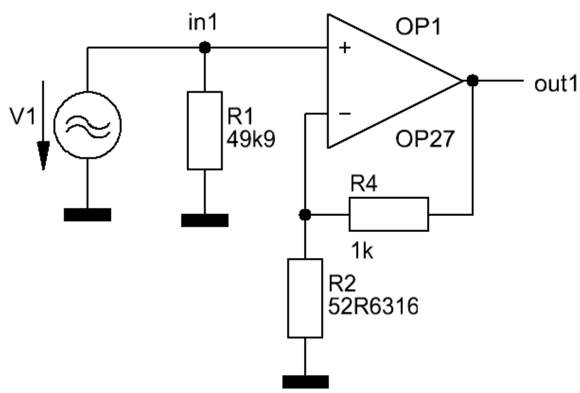

Fig. 2 Test circuit of an OP27 driven amp in non-inverting configuration

In contrast to the Fig. 1a result, Fig. 2 shows a not amusing difference between the calculated and simulated input referred noise voltage density “V(inoise)”, despite the fact that the output referred noise voltage density equals the calculated one.

The results here: In case of non-inverting amp arrangements LTspice uses as "gain" the following - in my eyes absolutely wrong - equation: signal gain (here = 1+R4/R2 = 20) minus 1 = 19 = "gain". A change of the "gain" command to "(gain+1)" does not correct the bug’s negative impact!

I also checked this approach with the more complex non-inverting amp example shown in **Fig. 3**. Again, I found a wrong "gain" of 99 = R6/R4, and not the correct gain, which is the signal gain 100 = R6/R4+1.

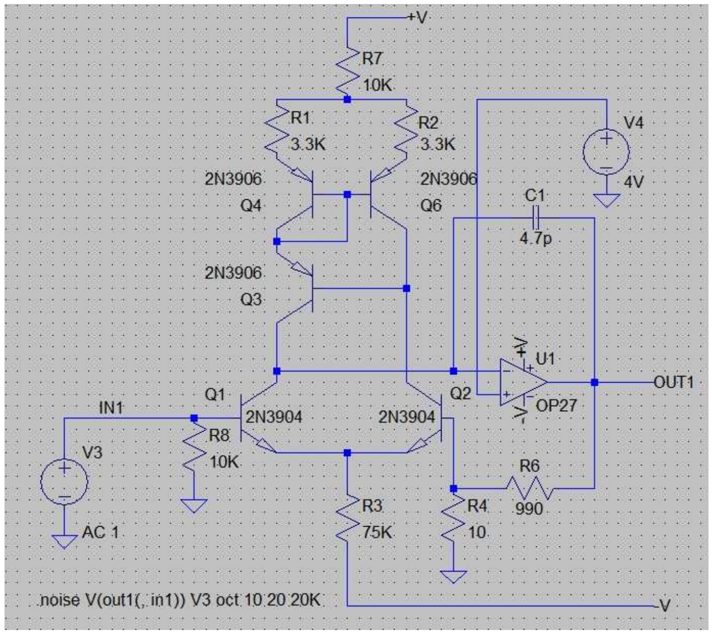

Fig. 3 Non-inverting example amp with a signal gain of 100

Higher amp gains will lead to less important errors, however, low amp gains will suffer much more from this kind of incorrect gain setting, eg. a gain of 2 will lead to an unacceptable error of 6dB at the amp's input. Hence, as shown in the plot of Fig. 4, the input noise voltage equals the output noise voltage after replacement of R2 with 1kΩ in Fig. 2.

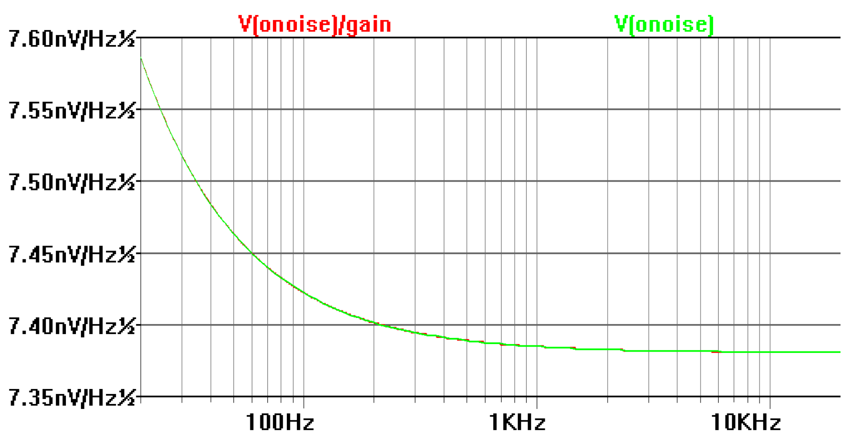

Fig. 4 Input referred noise voltage density trace (red & hidden) and output noise voltage density trace (green) of the Fig. 2 arrangement with a gain of 2

I hope that someone at LT will fix these strange gain application bugs asap!

During my various exercises, I found another interesting point to mention. Very often I use LT1028 op-amps and their LTspice noise characteristics fortunately are close to those in the LT data sheet. I also use OP27s and OP37s. According to the data sheets (LT and AD), at 1kHz their noise specs are given as 3.0nV/rtHz (selected version) or 3.2nV/rtHz (consumer version). In LTspice they show a dramatic improvement to 2.3nV/rtHz/1kHz (simulated with the **Fig. 5** -approach). Maybe it's a change of numbers. It should be corrected too.

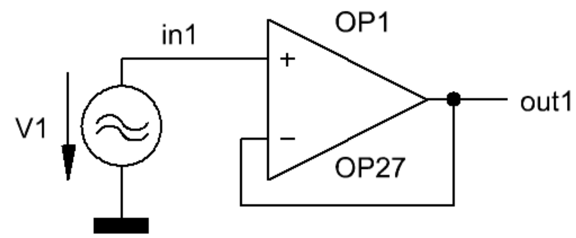

Fig. 5 Test circuit to get the input noise voltage of an op-amp via V(onoise)

To achieve a perfect noise analysis of analog amplification there are at least three additional points on my wish list (I hope I didn’t fail to notice them in the mass of possibilities):

1. In Fig. 1b the left ordinate of the .AC… analysis shows linear steps in dB only; it would be great if LT could implement an optional logarithmic ordinate without dB scaling.

2. It's easy to perform an .AC… analysis for a balanced amp. Taken from **Fig. 6** we can write the desired gain equation in the "Expression editor” for V(o1) [eg (V(o1)-V(o2)) / (V(i1)-V(i2))]. However, it is not easy (or even impossible) to find a way to perform either a noise analysis of different kinds of balanced or semi-balanced amplifiers (eg. bal in to bal out, bal in to se out, se in to bal out), or to identify 100% correlated parts of noise voltages that always exist in the balanced amp environment.

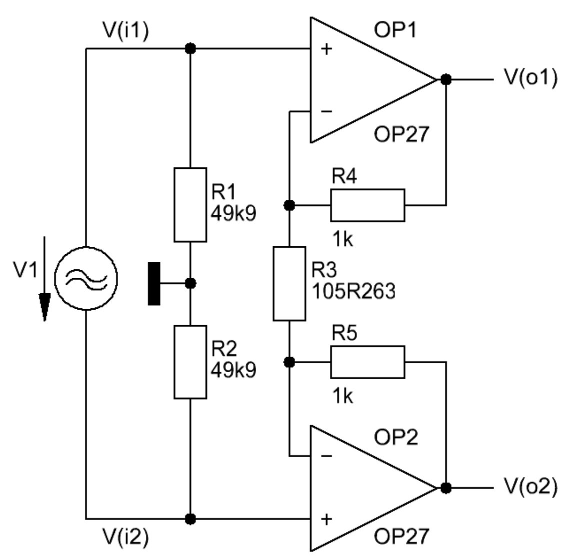

Fig. 6 Balanced amp configuration with a balanced signal gain of 20

3. I miss the possibility to simulate resistor excess noise. The Johnson noise represents only a portion of the total electronic noise that is generated by a DC potential across a resistor. That's why the simulated output noise voltage of Fig. 1b is questionable because - in addition to their Johnson noise - all resistors produce more or less excess noise, which is obviously not taken into account by LTspice.
   I tried to find a way to show the absence of excess noise and believe I found one with **Fig. 7a**. The output noise voltage density trace at the output o1 of the Fig. 7a test circuit

shows no deviation from the curve’s flatness between 10Hz and 100kHz (**Fig. 7b**). However, with a resistor noise index NI_e = -30dB the trace should look like the fully black one in Fig. 8.

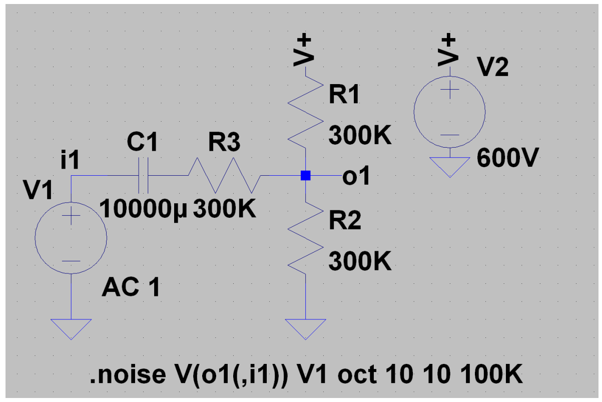

Fig. 7a Test circuit for the evaluation of resistor excess noise

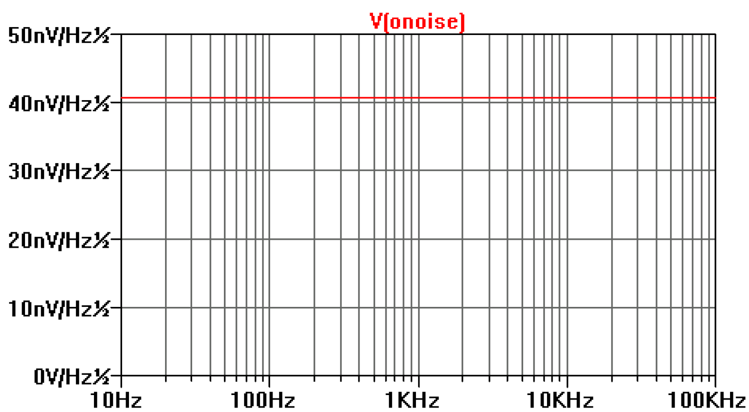

Fig. 7b LTspice created noise voltage density trace from the Fig. 7a output o1

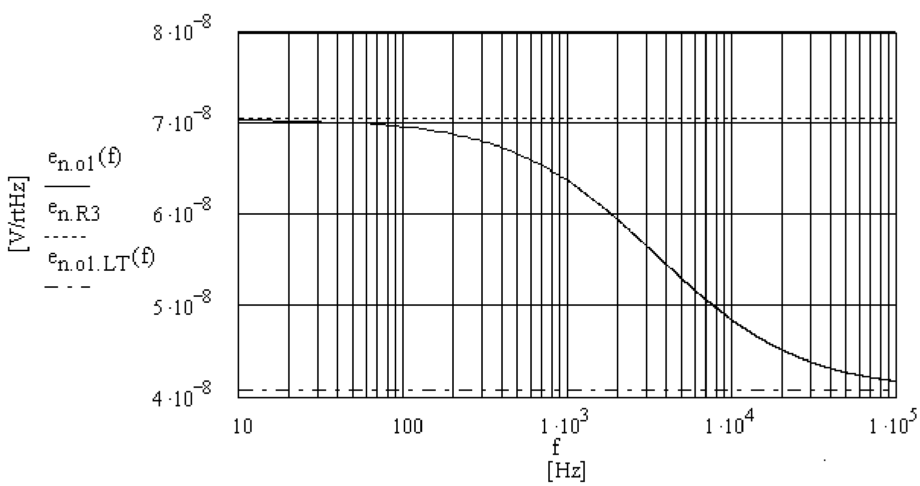

Fig. 8 Calculated noise voltage density traces of Fig. 7a

In **Fig. 8** the black solid trace shows the output noise voltage density of Fig. 7a’s R1||R2||R3 at the o/p o1, including excess noise of R1 and R2, the dash dotted line at the bottom represents the LT result of R1||R2||R3 without excess noise and the dotted line is the noise voltage of R3 alone, of course excluding excess noise.

The difference in calculated rms noise voltage in the range of 10Hz … 100kHz is not marginal:

- Fig. 7a excluding excess noise: 12.874µV
- Fig. 7a including. excess noise: 14.053µV
- Delta (rms): 5.641µV

After the above checks my overall conclusion does not change: as regards to electronic noise it's better trying to understand the noise behaviour of the DUT first, then calculate it with appropriate math, before struggling with bugs of imperfect simulation software, no matter if it's one to pay for or one for free. Nevertheless, I'll buy the book to learn more about this rather complex software.

Burkhard Vogel
Stuttgart, Germany