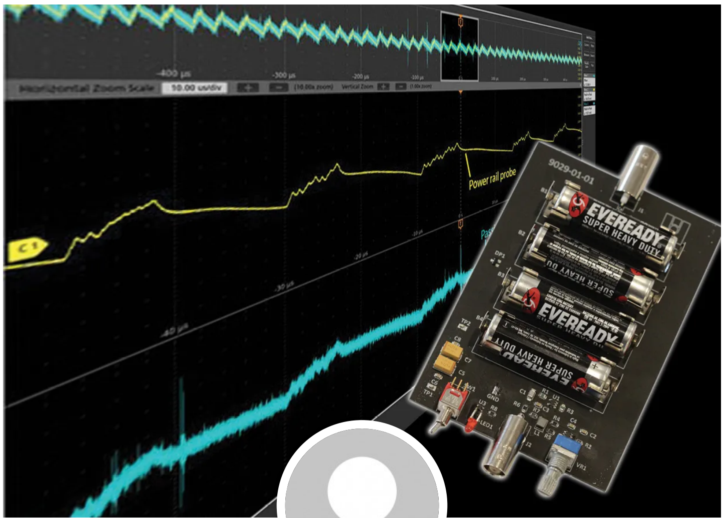
Inspiring the Evolution of Embedded Design.

Home | Magazine | NEWSLETTER | Research & Design Hub | Insights | Tech News | Resources | Community | Advertise

Projects • Research & Design Hub

Building a Power Rail Probe

02/01/2021



Written by Andrew Levido

An Exercise in Low-Noise Design

A power-rail probe is a special oscilloscope probe designed specifically to help probe DC power rails. Most of the larger oscilloscope manufacturers offer them, if you are prepared to shell out thousands of dollars. In this project article, Andrew sets out to see if he can build one with comparable performance, but for a lot less money.

Topics Discussed

Tech Used

hy would someone spend several

PROJECTS
Multi-Sensor Device Measures Water Pollution

more of three things: (1) look at the level of ripple and noise on the rail; (2) chase fast spikes or switching artifacts or (3) look at load transients such as over- or under-shoots, settling times and so forth. These are all usually millivolt-level signals riding on top of the DC rails, with periods ranging from nanoseconds to seconds.

You might try to use a standard passive probe with your oscilloscope switched to DC coupling, and rely on the vertical offset controls to center the waveform on screen. Unfortunately, you will find that at millivolt-per-division resolution, you are limited to just a volt or two of offset range, so you may not even be able to get a 3.3V rail on the screen, let alone a 12V rail.

If you flick the AC-coupling switch, you will be able to get the trace on the screen with millivolt resolution, but you will have just introduced a high-pass filter with a -3dB cut-off at a few tens of Hertz. You therefore will not be able to make much sense of load transients with a time constant of longer than a few tens of milliseconds.

ADC NOISE

If we are interested in measuring the noise on the power rail, the displayed noise will include that introduced by the oscilloscope's analog-to-digital converter (ADC), your chosen attenuation path and the input probe. There is not much you can do about the ADC noise (except buy a more expensive oscilloscope), but the attenuation path and probe noise are things we can control. With most oscilloscopes, the lowest noise is achieved when you switch in the 50Ω input termination. This is not practical with power rail measurements, since 50Ω would impose a significant load on the power supply, and the 50Ω input is usually only rated to ±5V to limit the power dissipation in the termination resistor.

A dedicated power rail probe is designed to overcome these limitations. It presents a relatively high-impedance load to the power rail, and provides a means to inject a DC offset to the signal, so the DC coupling can be used with high vertical sensitivity. It should have a very wide bandwidth, from DC to at least the oscilloscope's bandwidth, be compatible with the 50Ω input and should introduce as little additional noise as possible.

The commercial units I could find seemed to be specific to each manufacturer's proprietary active probe interface, and were integrated into the oscilloscope's on-screen user interface. I wanted to see what would be involved in rolling my own version of a power rail probe, so I did a quick survey of the available units to come up with a basic specification for mine.

5. I set myself an ambitious target of the probe contributing no more than 10% beyond a typical oscilloscope noise floor.

DESIGN

This design turned out to be an interesting exercise in low-noise design, and included a little bit of AC circuit analysis into the bargain. I think it might be interesting to walk through the highlights of the design process step by step.

I did not think a low-noise active circuit with a bandwidth in the gigahertz range was a practical proposition. Instead, I looked at splitting the probe into two parallel signal paths—an active, low-frequency path that deals with the DC offset, and a passive, high-frequency path. I started the design with the low-frequency path. This can be an ordinary summing amplifier followed by an inverting stage to restore the polarity of the output signal, as shown in the schematic (**Figure 1**).

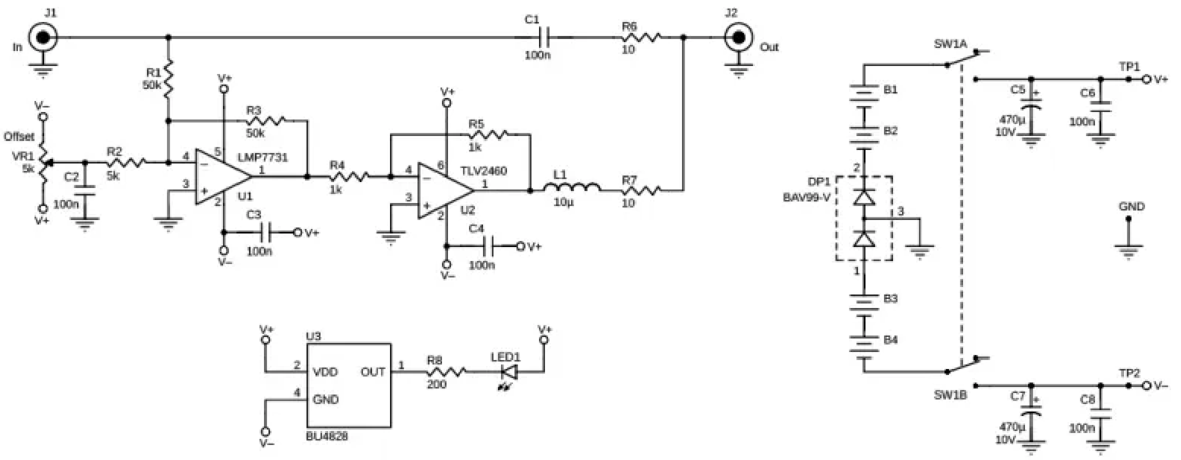


FIGURE 1 – Full schematic for the power rail probe. The DC path is a simple summing amplifier followed by an inverting stage. The AC path is a direct connection. The two paths are combined via C1 and L1. R6 and R7 ensure a flat frequency response.

This circuit pretty much designs itself, because we know the input impedance must be 50kΩ, and the feedback resistor R2 must be the same for a gain of -1. If we assume a split 5V supply (±2.5V), then we can choose R2 to be 5kΩ for a gain of -10, giving an offset range of ±25V, which is nicely in line with our specification. Since we are connecting R2 directly to the wiper of the potentiometer, the input impedance we will see on the offset summing input will only be 5kΩ at the extremes of the potentiometer.

In the middle, the impedance will be 5kΩ plus the two "halves" of the potentiometer resistance in parallel, reducing the gain from -10 to something less than that. For the 5kΩ pot I used, the gain of the offset branch is reduced to -8 at the center point. This means we will have slightly increased sensitivity at lower offset voltages. This is by no means a bad thing, and the alternative of adding a buffer between the pot and R2 is not desirable, because it will add more noise to the

PROJECTS

Multi-Sensor Device Measures Water Pollution

select. This also means we do not need to add resistors in the non-inverting inp amps, and can connect them directly to our circuit ground. Similarly, for the inv we just select R4 = R5 for a gain of -1. We will choose these values a little later, s affect the noise performance of our circuit.

We can now look at the high-frequency path. Since we don't have to worry about the DC offset, this path is just a direct connection between the input terminal and C1. **Figure 2a** shows a simplified view of how the low-frequency active path and the passive high-frequency path come together. The capacitor C and the load impedance R_L form a high-pass RC filter, and the inductor L similarly forms a low-pass RL filter with R_L. This is functionally equivalent to **Figure 2b**, which you may recognize as a classic LC notch filter. A steep notch appears in the frequency response at the resonant frequency given by:

$$f_0 = \frac{1}{2\pi\sqrt{LC}}$$

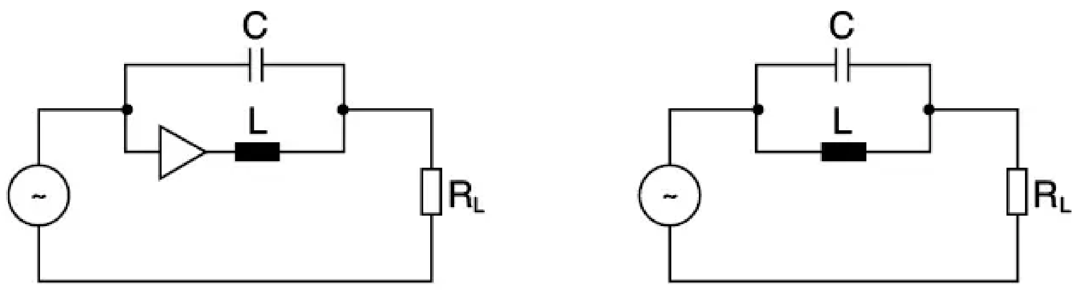
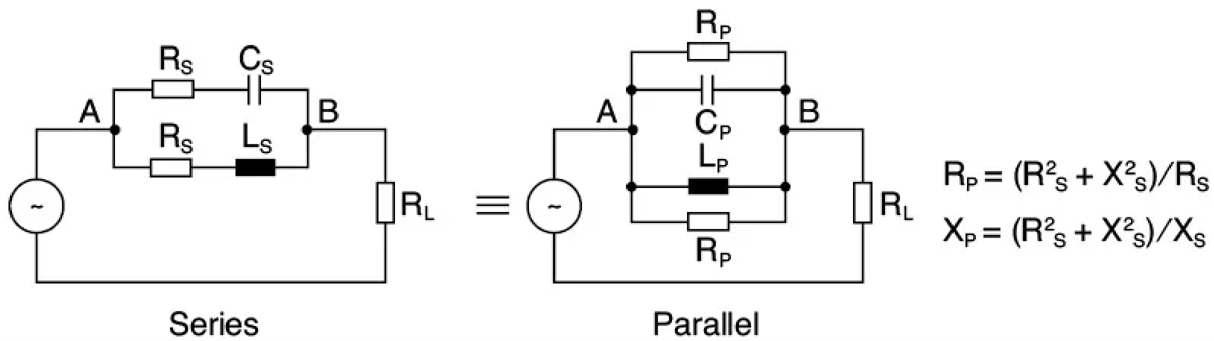


FIGURE 2 – The circuit on the left (a) is a simplified view of how the low and high frequency paths combine. The functionally equivalent circuit on the right (b) is instantly recognizable as a classic LC notch filter.

The depth of the notch is determined by the Q of the filter. Naturally, we do not want this notch in our frequency response, so we need to lower the Q of the filter by adding some series resistors R_S to the circuit, to provide some damping (**Figure 3**). We also need to preserve the symmetry of the branches, so each branch must have the same resistor value.



frequency response? The answer is not immediately obvious, but we can make exploiting the series-parallel transformation property of complex impedances. That for every series combination of real and imaginary impedances, there is a parallel combination that will have the same overall behavior when viewed fron Figure 3 shows how this looks for our circuit. The series $C_S + R_S$ and $L_S + R_S$ combinations become parallel C_P || R_P and L_P || R_P combinations.

At DC, our series circuit will look like an impedance of R_S, since the capacitor C_S is essentially open, and the inductor L_S a short. Similarly, at very high frequencies, the series circuit will also look like an impedance of R_S, because the inductor L_S will be essentially open and the capacitor C_S essentially a short. It is not obvious from the series circuit what the impedance will be at resonance.

This is where we need to look at the equivalent parallel circuit. We can see here that at resonance, when the L_P || C_P combination is presenting an infinite impedance, the end-to-end impedance will be the parallel combination of the two resistors. If we want the gain to be flat across the spectrum, this should also be equal to R_S. Substituting $R_P = 2R_S$ into the series-parallel transformation equation for R_P given in Figure 3, and using the expression above for the resonant frequency, a little algebra delivers the expression:

$$R_S = \sqrt{\frac{L_S}{C_S}}$$

Now we can choose some values. Let's start by choosing C to be 100nF, which doesn't seem like too much of an additional capacitive load, given that power rails are generally bypassed by many 100nF capacitors. A value of 100nF will form an RC high-pass filter with a corner frequency of around 31.8kHz.

We want to choose the inductor L such that the corner frequency of the RL low-pass filter is considerably higher than this, but not so high that we create an unnecessary op amp design challenge. Choosing 1MHz, for example, gives us an L value of 7.9µH. Rounding this up to a sensible value like 10µH gives a corner frequency of around 796kHz, which should work fine. We can now finally calculate R_P, which turns out to be 10Ω—a nice, round value that gives us an overall gain through the probe of 1:1.2, exactly our design brief.

NOISE 101

Before we can we can select the op amps for the circuit, let's look at the noise issues involved. This op amp selection will be driven largely by the need to add the lowest possible amount of

temperature will develop a noise voltage across its terminals of 1.3µV_{RMS}, if mea
voltmeter with 10kHz bandwidth.

This brings us to an important point about noise. If we want to calculate a noise
current, we must always specify the bandwidth over which we are concerned. Because we don't always know what the band of interest is when specifying components, we tend to talk about noise densities, which is a measure of the noise voltage or current per unit of bandwidth. Noise densities are expressed in terms of voltage or current per square root Hertz. You just multiply the voltage or current noise density by the square root of the bandwidth to get the root mean square (RMS) voltage or current.

While we are on the subject of noise, another important thing to keep in mind is how noise in a circuit adds. Since noise is random, as long as the noise sources are not correlated, the two sources don't add arithmetically, but rather, as the square root of the sum of squares:

$$e_n = \sqrt{e_{n1}^2 + e_{n2}^2 + \ldots}$$

With these two points covered, let's get back to Johnson noise. The voltage noise density of Johnson noise in a resistor is given by:

$$e_n = \sqrt{4kTR}$$

where k is Boltzmann's constant and T is the temperature in Kelvin. At room temperature, this can be reduced to: $e_n = 127 \times 10^{-12}$ in units of "V/√Hz". We will use this equation later to calculate the noise produced by the resistors in our circuit.

The two other sources of noise are generally specific to semiconductors. Shot noise is the noise produced by the discrete nature of charge when charges are acting independently, as they do in semiconductors (but not in conductors). 1/f or flicker noise is an artifact of the materials used and methods of construction of the devices.

Johnson and shot noise are called "white noise," and have equal energy in every unit of bandwidth. Flicker noise is "pink" noise and has equal energies in every decade of bandwidth. If you plotted the energy of flicker noise versus frequency, we would see the same energy in the 1-10Hz decade as the 10-100Hz or 100Hz-1kHz decades. The amount of energy therefore falls off with the inverse of frequency—hence the 1/f name. In a semiconductor device such as an op amp, there will be a white-noise component that is flat with frequency, and a pink-noise component that will fall off at 1/f. The point where the 1/f noise has the same value as the white noise is designated f_c. This figure is used if the frequency of interest is low enough that 1/f noise becomes important—as we shall see later.

PROJECTS
Multi-Sensor Device Measures Water Pollution

As shown in **Figure 4**, the noise model of an op amp looks like a noise voltage sc with the op amp input, and noise current sources i_n driving current out of the ir These currents will create noise voltages across the source resistance as seen b terminal. There will also be a Johnson noise component in the source resistance that appears at the op amp output terminal will be the sum of these three sources multiplied through the op amp gain. If there is noise on the op amp power rails, this too will affect the overall noise, as seen at the op amp output. I will ignore this for now, but check whether this is reasonable once we have designed the power supply.

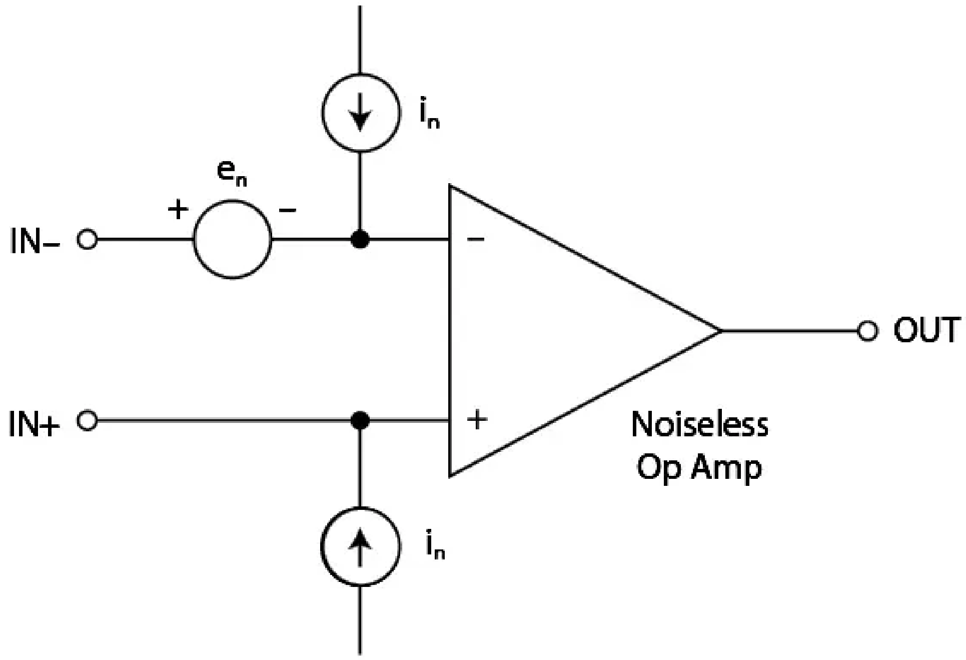


FIGURE 4 – The noise model of an op amp has a voltage noise source in series with the input, and sources forcing current noise out of the input pins. Op amp noise is specified in terms of these noise densities and the frequency f_c, at which the flicker noise begins to dominate.

COMPONENT SELECTION

We now need to choose the op amps. In this case our primary requirement is for low noise. We also need a bandwidth well above the 796kHz corner frequency of the LC filter—say 5MHz or higher. We want to operate from low voltage supply (±2.5V), and U2 needs to be able to drive a volt into a 50Ω load.

One candidate is the Texas Instruments (TI) LMP7731. This has a GBW product of 22MHz, swings almost to the rails with a 5V supply, and has excellent noise figures with e_n = 2.9nV/√Hz, i_n = 2.2pA/√Hz and f_c = 3Hz. Quiescent current is 4mA, and it costs less than $2. The only shortcoming is that its output drive capability is not enough. Another candidate is TI's TLV2460.

the LMP7731 for the summing amp.

We can calculate the noise in the summing amplifier by adding up the various n shown in **Figure 5**. At the inverting input of U1 we will see three sources of nois voltage noise, the amplifier current noise reflected across the source resistance, and the Johnson noise of the source resistance itself. The source resistance is simply the parallel combination of R1, R2 and R3, which is around 4.2kΩ. Because the gain of the stage is 1, the output voltage noise will be same as the input noise. Remembering that we add noise as the root sum of squares, the total voltage noise density at the summing amplifier output can be calculated to be 12.7nV/√Hz.

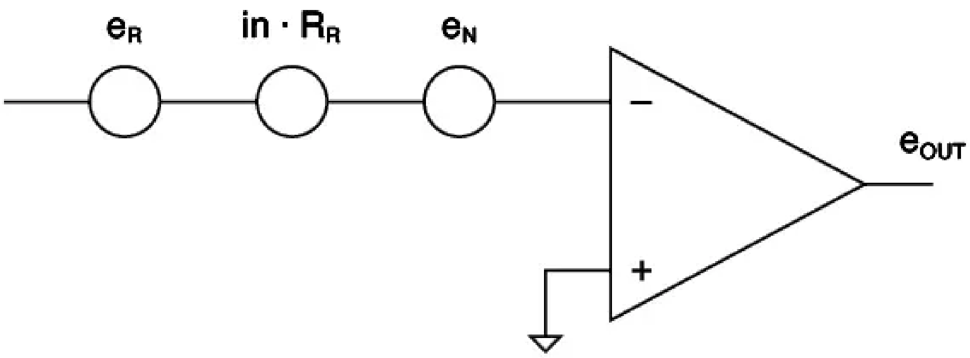


FIGURE 5 – When calculating the noise in an inverting amplifier, we add the root sum of squares of the amplifier input noise voltage, the voltage produced by the input noise current in the source resistance, and the Johnson noise produced by the source resistance itself—all multiplied through the op amp stage gain.

The calculation for the second stage is similar, but this time we also have to add the noise produced by the first stage. In this case, we will choose the resistors R4 and R5 to each be 1kΩ, so the source resistance is only 500Ω, thus reducing the noise in this part of the circuit. Again, the gain of the stage is 1, so the output noise will equal the input noise. Doing the math gives us a voltage noise density of 17.0nV/√Hz. The only other contributor to the low-frequency path noise will be the 10Ω resistor, which will contribute just 0.4nV/√Hz. This is negligible and can be ignored. Note that we can't make this simplification in the high-frequency path, as we shall see later.

NOISE AT OUTPUT

Now, to calculate the noise voltage at the output of the low-frequency path, we need to specify a bandwidth. The lower limit is DC, so that is easy. You might think the upper limit should be the -3dB bandwidth of the LC filter (796kHz), but the reality is that the LC filter is far from a brick-wall filter, so there will be noise energy at higher frequencies. It turns out that we must adjust this upper frequency by a factor of π/2 to account for the gradual roll-off. This means our upper frequency bound is 1.25MHz. Crunching the numbers gives a noise voltage of 19.0μV_{RMS}.

$$v_n = e_n\sqrt{(f_h - f_l + f_c In(f_h / f_l))}$$

Instead of just multiplying the noise density by the square root of bandwidth $f_h - f_l$ we now have an extra logarithmic term that accounts for the 1/f noise. For obvious reasons, we can't plug 0Hz into this equation for the lower limit. So, we choose a suitably low frequency, say 1µHz, and again use 1.25MHz for the upper limit. This gives us a revised noise voltage of 21.2µV which just tells us that the 1/f noise contribution is small but not negligible.

Finally, we need to calculate the noise voltage due to the high-frequency path. In the same way as before, we need to extend the lower frequency cut-off from the -3dB value of 31.8kHz by a factor of π/2 to 20.3kHz. The upper bound in frequency is our design limit of 1GHz. The only contributor to this noise is the 0.4nV/√Hz Johnson noise in the 10Ω resistor. Calculating this gives a voltage noise in the high-frequency path of 22.5μV_{RMS}—not too different from that for the low-frequency path.

If we add these two together, using the root of the sum of squares, of course, we get a final noise figure of 30.9μV_{RMS}. The noise floor of a good oscilloscope will be in the order of 100µV – 300µV at 1mV per division. Adding our measurement noise to that of the lower end of the range will increase the oscilloscope noise floor to 105µV, well inside the 10% figure we set ourselves.

POWER SUPPLY

The op amps we have chosen work over a voltage range of 1.8 to 5.5V and 2.7 to 6.0V, respectively. The TLV2460 must be able to drive 20mA into 50Ω, at which point it can swing to ±400mV of the rails, so we need a minimum supply of ±1.4V and a maximum of about ±2.5V. Adding up the quiescent and load currents of the two op amps, we reach a total of about 25mA. The alkaline AA cells I looked at had a capacity of 3,000mA-hours at 25mA, for a nominal life of 120 hours, which should be enough.

Two cells in series for each rail would give a voltage range of ±3.0 to ±2.0V over the discharge life of the battery. Adding a diode drop to each rail will reduce this range to nominally ±2.4 to ±1.4V, which is pretty close to what we need. The only downside is that at low battery voltages, the maximum offset we will be able to inject will reduce to ±14V. If this is a problem for you, you can adjust the value of R2.

The noise calculations above assumed a perfectly noiseless power supply. If there were noise in the supply, it would appear at the amplifier outputs, attenuated by the op amp's power supply rejection ratio (PSRR). This is typically in the order of -100 dB at low frequencies, but gets a lot worse at high frequencies. The ops amps we chose have a PSRR of around -35dB at 1MHz.

lights up. This embellishment is not strictly necessary, but does make for a mor instrument.

CONSTRUCTION AND RESULTS

I laid out a simple, two-layer PCB as shown in **Figure 6**. The high-frequency path is a straight 50Ω controlled-impedance trace, running down the center of the board under the batteries. The rest of the layout is not super critical. It is important, however, to enclose the circuitry in a well shielded case to prevent stray noise from being coupled into the circuit. I used a shielded aluminum case from Hammond Manufacturing. It was the most expensive component in the project.

FIGURE 6 – The finished PCB assembly. The input is at the top, with the output and controls at the bottom. The whole assembly slides into an aluminum RF screened case (not shown).

To use the probe, simply connect to your oscilloscope via a 50Ω BNC-to-BNC cable, and switch to 50Ω input termination. For the best results when using the probe, it should be connected to the

So, how did it work out in practice? Real-life components have tolerances and p
physical construction also influences their actual performance. Measuring the g
decades of frequency shows a fairly flat response from DC to just under 100MH
limit of my measurement capabilities (**Figure 7**).

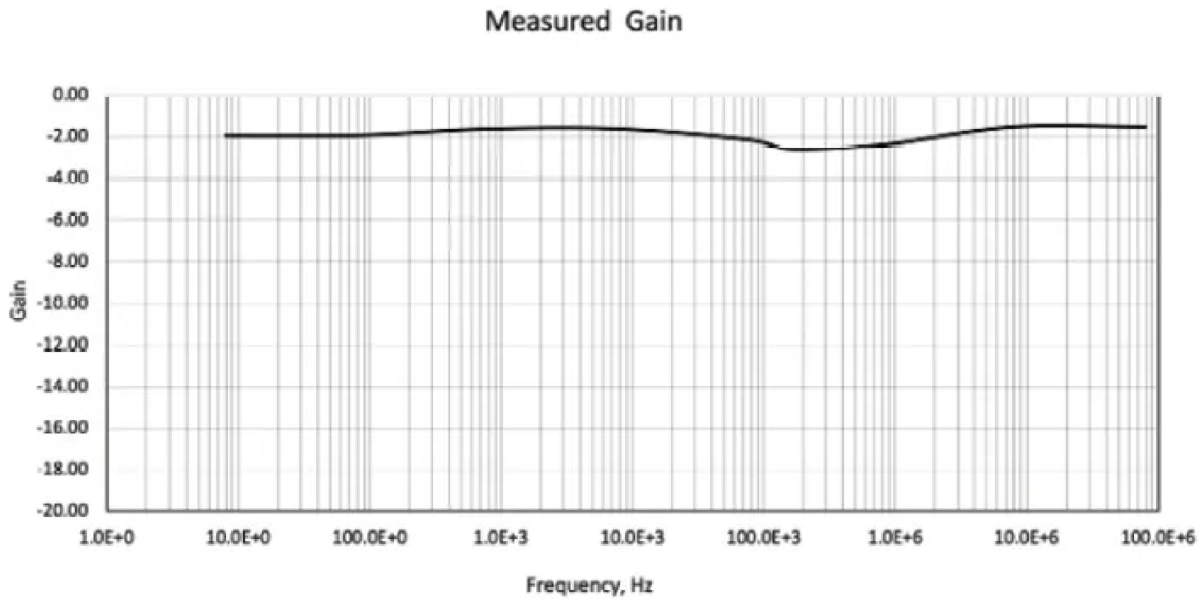


FIGURE 7 – Measured gain over the frequency range I could measure with my setup. The gain is flat to within 1dB. The slight dip at the resonant frequency of around 160kHz is due to the non-ideal nature of real-life components.

There is a dip centered on 160kHz, which tells us that the matching of the damping resistors to the capacitive and inductive impedances is not perfect. This error is less than 1dB at its worst point, just on the design target. Given that I used standard components, this is not a bad result.

The noise floor of my 200MHz oscilloscope in the 1mV-per-division range is around 155μV_{RMS}, and adding the power rail probe increases this to around 200μV_{RMS}—noticeably worse than my calculations suggest, but not catastrophic. If I switch-in the 20MHz bandwidth limit, the noise floor drops to 96μV_{RMS}, and with the probe connected, the noise increases to 108μV_{RMS}, which is much more in line with what I expected.

This suggests the noise is mostly in the 20-200MHz range, and most likely relates to the imperfect shielding of the enclosure. I might try to address this in a future iteration of the design, but to be frank, I am happy enough with this result, given the type of work I do, and the fact that total cost of parts is well under $100.

RESOURCES

Texas Instruments

Hammond Manufacturing
Hammond 1457J1201EBK Shielded Aluminum Encloser 120mm x 84mm x 28mr

Hammond Manufacturing | www.hammfg.com
Rohm Semiconductor | www.rohm.com
Texas Instruments | www.ti.com

PUBLISHED IN CIRCUIT CELLAR MAGAZINE • OCTOBER 2020 #363 – Get a PDF of the issue

Andrew Levido

+ posts

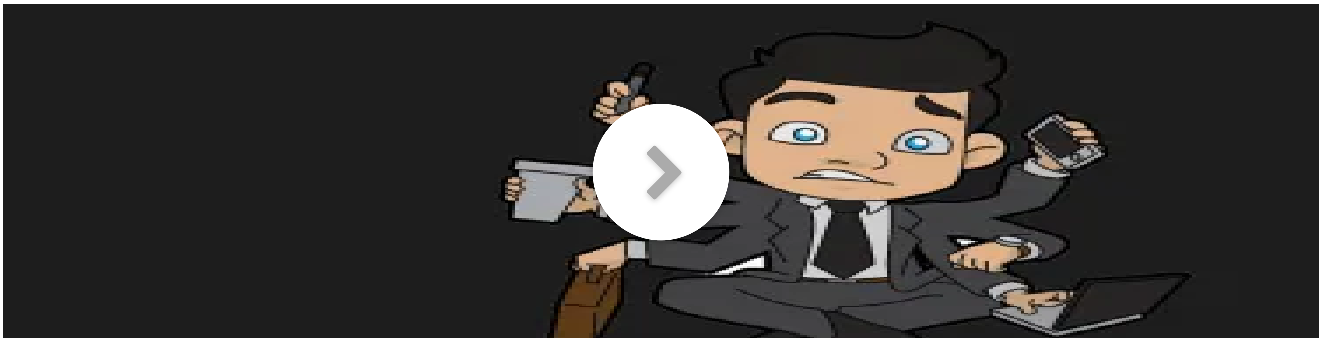
Andrew Levido (andrew.levido@gmail.com) earned a bachelor's degree in Electrical Engineering in Sydney, Australia, in 1986. He worked for many years in R&D for power electronics and telecommunication companies, before moving into management roles. Andrew has maintained a hands-on interest in electronics, particularly embedded systems, power electronics and analog design. Over the years, he has written articles for various electronics publications, and provides consulting services as time allows.

#ADC #op amp #oscilloscope #power rail probe #voltage detector

PROJECTS

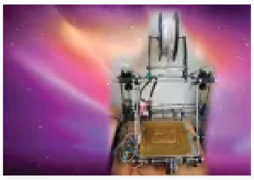
Multi-Sensor Device Measures Water Pollution

Multi-Sensor Device Measures Water Pollution



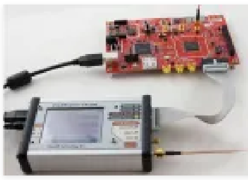
FreeRTOS – Part 2

You may also like



Projects • Research & Design Hub

Stepper Motor Emulator (Part 2)



Design Solutions • Research & Design Hub

Power Analysis of ECC Hardware Implementations

Projects • Research & Design Hub

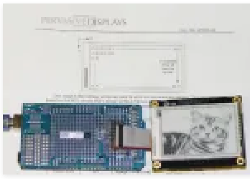
Controlled Circuit Breaking

Announcement • Research & Design Hub

June Circuit Cellar: Sneak Preview

Research & Design Hub • Tech Trends

Embedded AI Designs Embrace FPGAs and GPUs



Projects • Research & Design Hub

Powerless Display Technology (Part 1)

PROJECTS

Multi-Sensor Device Measures Water Pollution

Subscription Help
◊ Back Issue (PDFs)

◊ Subscribe