Analog Electronics | Volume 29

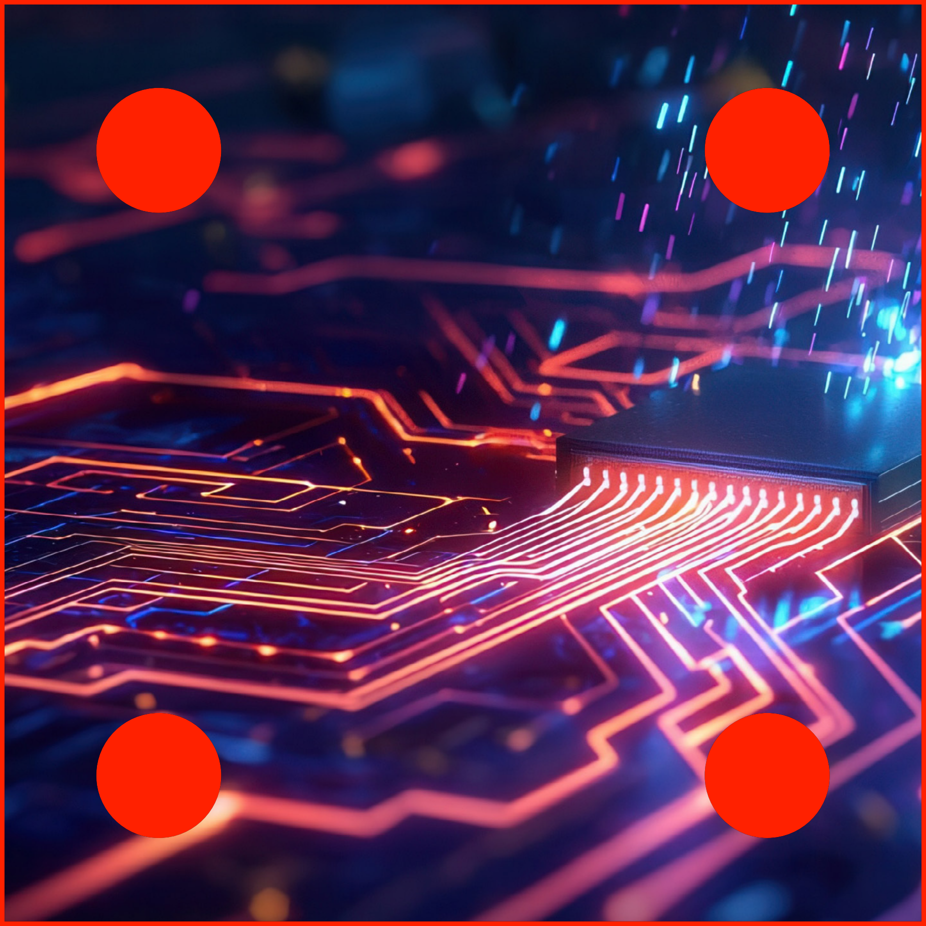
We get technical

Analog integrators: How to apply them for sensor interfaces, signal generation, and filtering

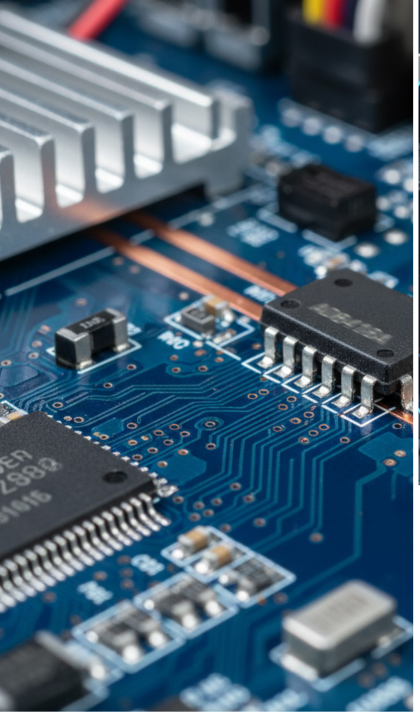
Efficiently implement current monitoring using integrated bidirectional current sense amplifiers

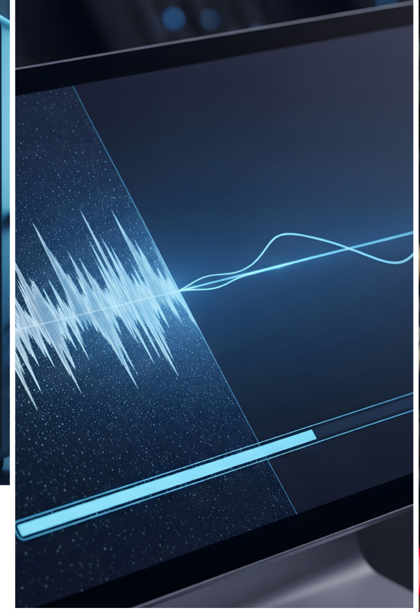
Can an ADC be just a random number generator?

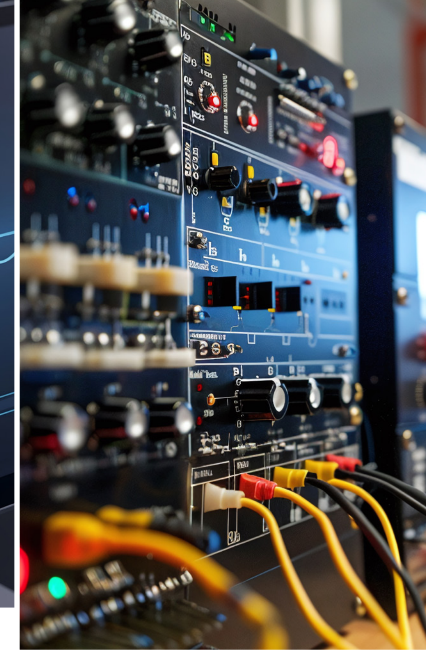
Analog fundamentals: How sample and hold circuits work and ensure ADC accuracy



DigiKey







contents

Editor's note

Welcome to the DigiKey eMagazine Volume 29 – Analog Electronics.

This edition is all about getting comfortable with the analog and mixed-signal building blocks that show up in everyday designs. Whether you're brushing up on the basics or looking for practical tips you can use right away, the articles in this volume focus on understanding how things work—and how to make them work better.

We start with core concepts like integrators and sample-and-hold circuits in *"Analog Integrators: How to Apply Them for Sensor Interfaces, Signal Generation, and Filtering"* and *"Analog Fundamentals: How Sample and Hold Circuits Work and Ensure ADC Accuracy."* From there, several articles dive into filtering and signal conditioning, including *"The Basics of Low and High-Pass Filters," "Signal Processing: Exponentially Moving Average (EMA) Filter,"* and *"Improve Test Performance with Low-Cost Signal Sources Using Inline Filters."* You'll also find a practical look at measurement techniques in *"Efficiently Implement Current Monitoring Using Integrated Bidirectional Current Sense Amplifiers,"* along with a fun and slightly unconventional take in *"Can an ADC Be Just a Random Number Generator?"*

Altogether, this collection is meant to be informative, approachable, and useful—whether you're designing a new system, troubleshooting an old one, or just curious about how analog circuits really behave. We hope you enjoy the read and find something you can put to work in your next project.

Analog integrators: How to apply them for sensor interfaces, signal generation, and filtering

By Art Pini
Contributed By DigiKey's North American Editors

Before the electronics world went digital, control systems, which are based on the solution of differential equations, used analog computation to solve those equations. As a result, analog computers were quite common as almost all solutions to differential equations required the ability to integrate signals. While control systems have mostly gone digital and numerical integration has replaced analog integration, there is still a need for analog integrator circuits for the operation of sensors, signal generation, and filtering. These applications use integrators, based on operational amplifiers (op amps) with capacitive elements in the feedback loop, to provide necessary signal processing in low-power applications.

Though still important, many designers may easily overlook their utility. This article provides an overview of integrator circuits and guidance on proper design, component selection, and best practices to achieve excellent performance using several examples from Texas Instruments.

Basic inverting integrator

The classic analog integrator uses an op amp with a capacitor as a feedback element (Figure 1).

The output voltage, V_{OUT}, of the integrator as a function of the input voltage, V_{IN}, can be calculated using Equation 1.

$$V_{OUT} = -\frac{1}{RC}\int V_{IN}dt$$

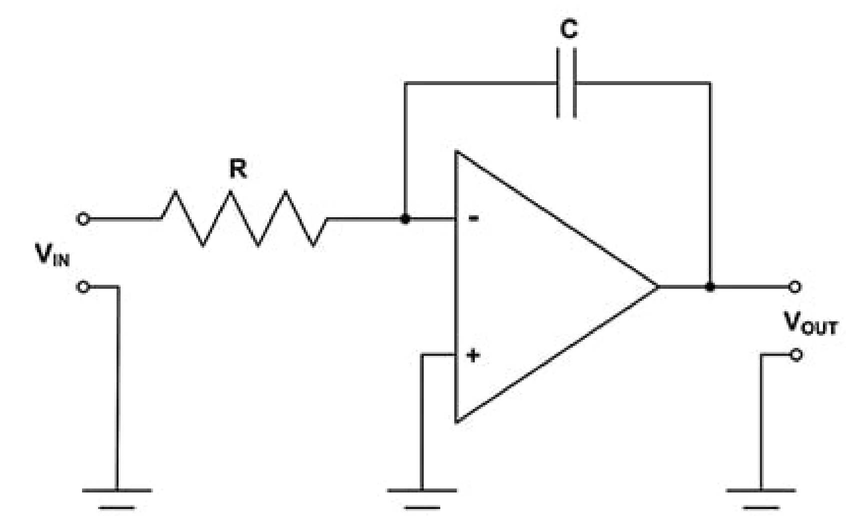


Figure 1: The basic inverting analog integrator consists of an op amp with a capacitor in its feedback path. (Image source: DigiKey)

The gain factor of the basic inverting integrator is -1/RC applied to the integral of the input voltage. In practice, capacitors used for integrators should have tolerances less than 5% and low temperature drift. Polyester capacitors are a good choice. Resistors with ±0.1% tolerance should be used in critical path locations.

There is a limitation in this circuit in that at DC the capacitor represents an open circuit and the gain goes to infinity. In a working circuit, the output would rail, going to either a positive or negative power supply rail depending on the polarity of the non-zero DC input. This can be corrected by limiting the DC gain of the integrator (Figure 2).

Adding a high-value resistor (R_F) in parallel with the feedback capacitor limits the DC gain of the basic integrator to the value of $-R_F/R$, resulting in a practical device. This addition solves the DC gain issue but does limit the frequency range over which the integrator works. Looking at a real circuit is helpful in understanding this limiting (Figure 3).

This circuit uses a Texas Instruments LM324 op amp. The LM324 is a good general purpose op amp with low input bias current (45 nanoamps (nA) typical), low offset voltage (2 millivolts (mV) typical), and a gain-bandwidth product of 1.2 megahertz (MHz).

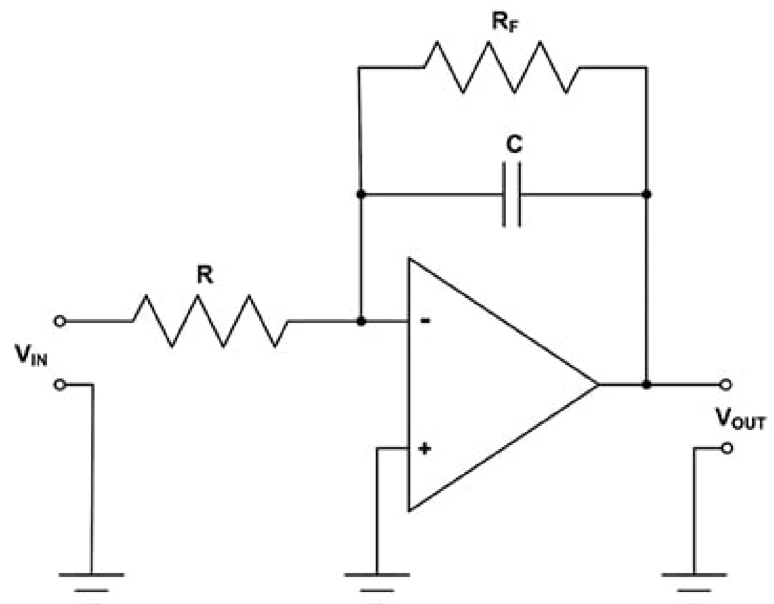


Figure 2: Adding a large resistor in parallel with the feedback capacitor limits the DC gain and results in a practical integrator. (Image source: DigiKey)

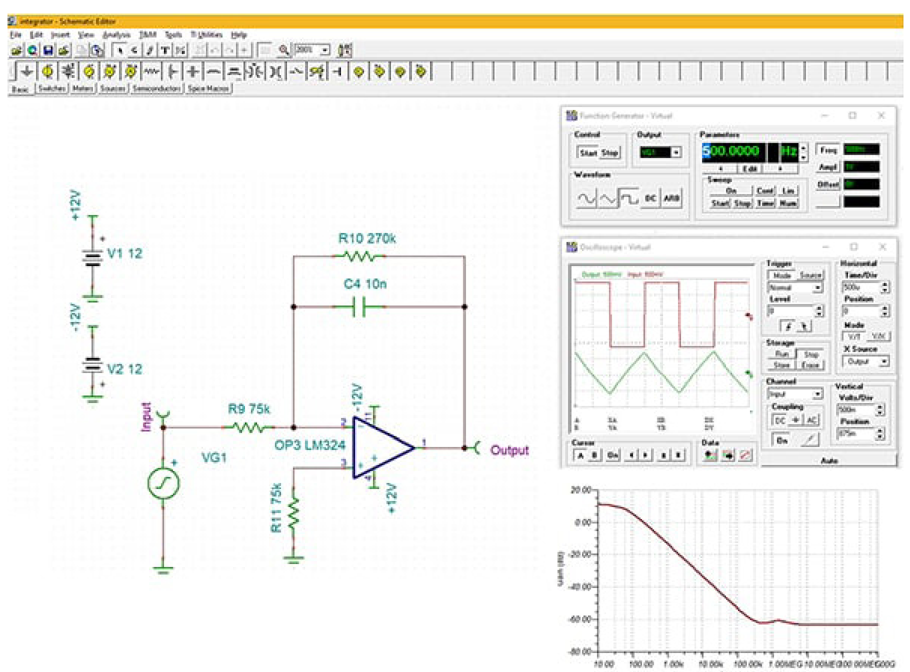


Figure 3: A TINA-TI simulation of a practical integrator using real components. (Image source: DigiKey)

The circuit input is driven by the simulator's function generator with a 500 hertz (Hz) square wave. This is shown as the upper trace on the simulator's oscilloscope. The circuit integrates the square wave and the output is a 500 Hz triangle function shown as the lower oscilloscope trace.

The DC gain is -270 kilohms (kΩ)/75 kΩ or -3.6 or 11 decibels (dB); this is seen in the transfer function of the circuit, shown in the lower right grid in Figure 3. The frequency response rolls off at -20 dB per decade from about 100 Hz to about 250 kilohertz (kHz). This is the useful frequency range of integrator operation and it is related to the op amp gain-bandwidth product.

A newer op amp is the Texas Instruments TLV9002. This 1 MHz gain-bandwidth amplifier has an input offset voltage of ±0.4 mV and an extremely low bias current of 5 picoamps (pA). As a CMOS amplifier, it is intended for a wide range of low-cost portable applications.

It's important for designers to keep in mind that an integrator is a cumulative device. As such, and without appropriate compensation, the input bias current and input offset voltage can result in the capacitor voltage increasing or decreasing over time. In this application the input bias current and offset voltage are relatively low, and the input voltage forces the feedback capacitor to discharge periodically.

In applications that use the accumulation functionality, as when measuring charge, there has to be a mechanism for resetting the voltage and establishing initial conditions in the integrator. The Texas Instruments ACF2101BU has such a mechanism. It is a dual switched integrator that incorporates a built-in switch to discharge the feedback capacitor. Since the device is intended for applications requiring charge accumulation, it has an extremely low bias current of 100 femptoamps (fA) and a typical offset voltage of ±0.5 mV.

A similar switched integrator/transimpedance amplifier is the Texas Instruments IVC102U. It is intended for the same range of applications as the ACF2101BU but differs in being a single device per package. It also has three internal feedback capacitors. It incorporates switches to discharge the capacitor bank and to connect the input source so that the designer has the ability to control the integration period and include a hold operation, as well as discharge the voltage on the capacitor.

Non-inverting integrator

The basic integrator inverts the integral of the signal. While a second inverting op amp connected in series with the basic integrator can restore the original phase, it is possible to design a non-inverting integrator in a single stage (Figure 4).

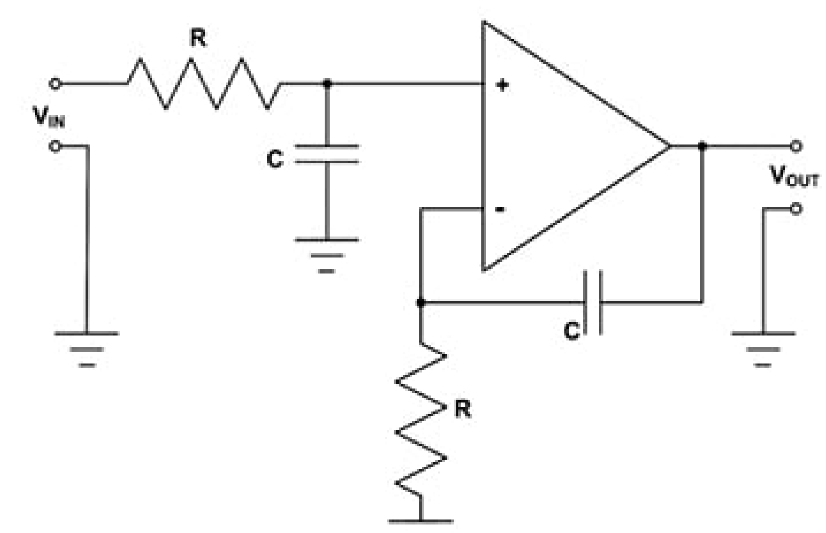


Figure 4: A non-inverting integrator based on a difference amplifier op amp configuration can ensure the output phase matches that of the input. (Image source: DigiKey)

The non-inverting version of the integrator uses a differential integrator to keep the output in phase with the input signal. This design adds additional passive components, which should be matched for optimum performance. The relationship between the input and output voltages is the same as the basic integrator with the exception of the sign, as shown in Equation 2:

$$V_{OUT} = \frac{1}{RC}\int V_{IN}dt$$

Other adaptations to the basic integrator can be realized using traditional op amp circuits. For instance, multiple voltage inputs (V_1, V_2, V_3, …) can be added by summing each through its own input resistor (i.e., R_1, R_2, R_3, …) to the non-inverting input of the op amp. The resultant output of this summing integrator is calculated

$$V_{OUT} = -\frac{1}{C}\int\left(\frac{V_1}{R_1}+\frac{V_2}{R_2}+\frac{V_3}{R_3}+\cdots\right)dt$$

using Equation 3:
If R1=R2=R3=R, then the output is

$$V_{OUT} = -\frac{1}{C}\int\left(\frac{V_1}{R_1}+\frac{V_2}{R_2}+\frac{V_3}{R_3}+\cdots\right)dt$$

calculated using Equation 4:
And the output is the integral of the sum of the inputs.

Some common integrator applications

Historically, integrators have been used to solve differential equations. For example, mechanical acceleration is the rate of change or derivative of its velocity. Velocity is the derivative of displacement. The integrator can be used to take the output of an accelerometer and integrate it once to read velocity. If the velocity signal is integrated, then the output is displacement. This means that by using an integrator, the output of a single transducer can produce three distinct signals: acceleration, velocity, and displacement (Figure 5).

The input from the accelerometer is integrated and filtered to obtain the velocity. The velocity is integrated and filtered to yield the displacement. Note that all the outputs are AC coupled. This eliminates having to deal with the initial conditions of each integrator.

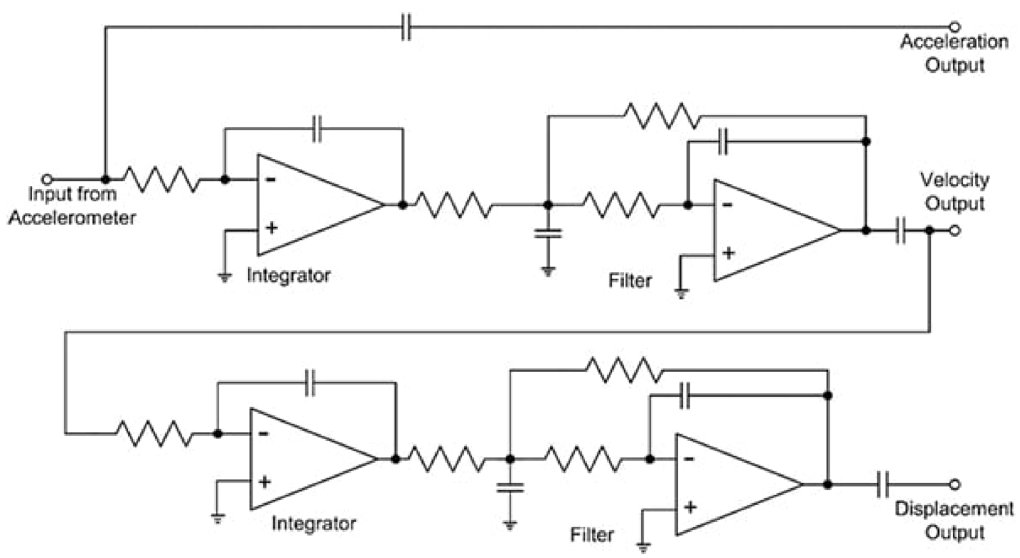


Figure 5: Using dual integrators, a designer can produce acceleration, velocity, and displacement readouts from an accelerometer. (Image source: DigiKey)

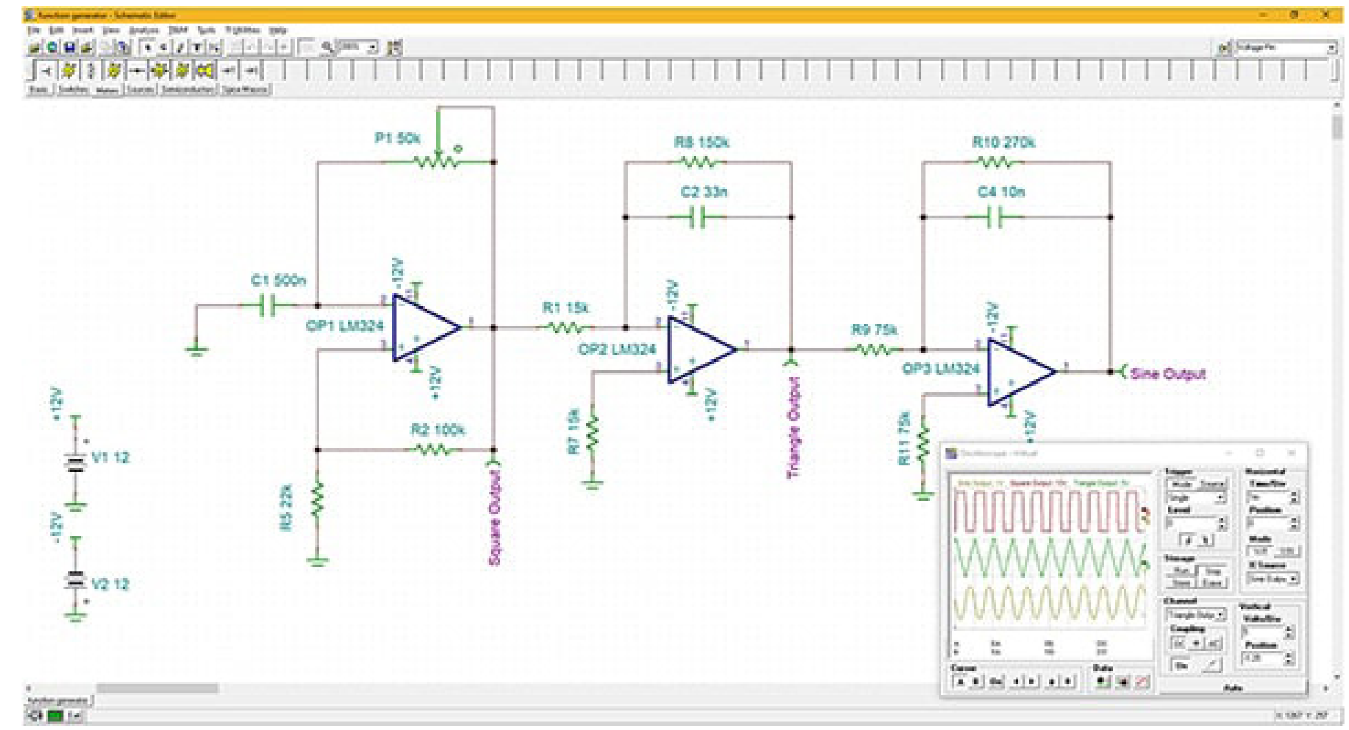


Figure 6: A function generator designed using three LM324 stages. OP1 is a relaxation oscillator generating a square wave; OP2 is an integrator that converts the square wave into a triangle wave; and OP3 is another integrator that operates as a low pass filter to remove the harmonics of the triangular wave, resulting in a sinewave. (Image source: DigiKey)

Function generator

Function generators, which output multiple types of waveforms, can be constructed with multiple integrators (Figure 6).

The function generator is designed around the LM324, which was discussed earlier as a practical integrator. In this design, shown as a TINA-TI simulation, three LM324 op amps are used. The first, OP1, is used as a relaxation oscillator and produces a square wave output at a frequency determined by C1 and potentiometer P1. The second stage, OP2, is wired as an integrator and converts the square wave into a triangle wave. The final stage, OP3, is wired as an integrator but is functionally a low pass filter. The filter removes all the harmonics from the triangle wave and outputs the fundamental frequency sinewave. The outputs of each stage appear in the simulator oscilloscope in the lower right of Figure 6.

Rogowski coils

Rogowski coils are a class of current sensors that measure alternating current sources using a flexible coil that is wrapped around the current-carrying conductor being measured. They are used for measuring high-speed current transients, pulsed currents, or 50/60 Hz line power.

Rogowski coils perform a function similar to a current transformer. The primary difference is that the Rogowski coil uses an air core as opposed to the ferromagnetic core used in a current transformer. The air core has a lower insertion impedance, resulting in a faster response and the absence of saturation effects when measuring large currents. The Rogowski coil is extremely easy to use (Figure 7).

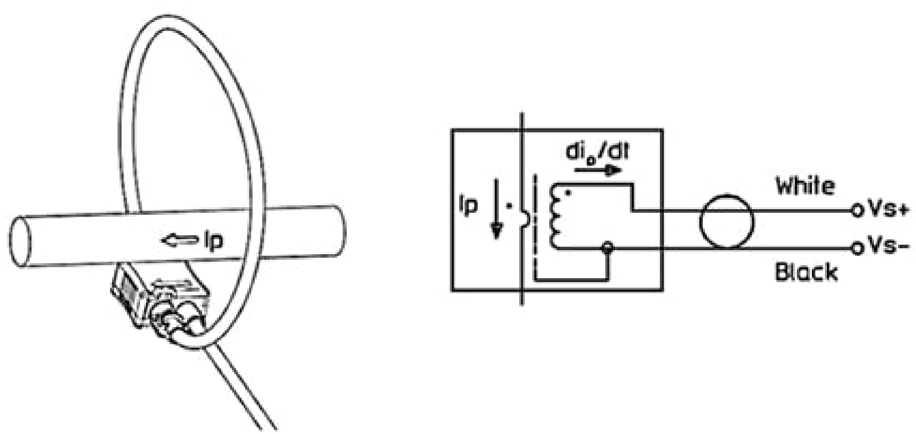


Figure 7: A simplified diagram showing the installation of a Rogowski coil about a current-carrying conductor (left) and the equivalent circuit for this setup (right). (Image source: LEM USA)

A Rogowski coil, like the [LEM USA ART-B22-D300](), is simply wrapped about the current-carrying conductor as shown on the left in Figure 7. The equivalent circuit of the Rogowski coil is shown on the right. Note that the output of the coil is proportional to the derivative of the measured current. An integrator is used to extract the sensed current.

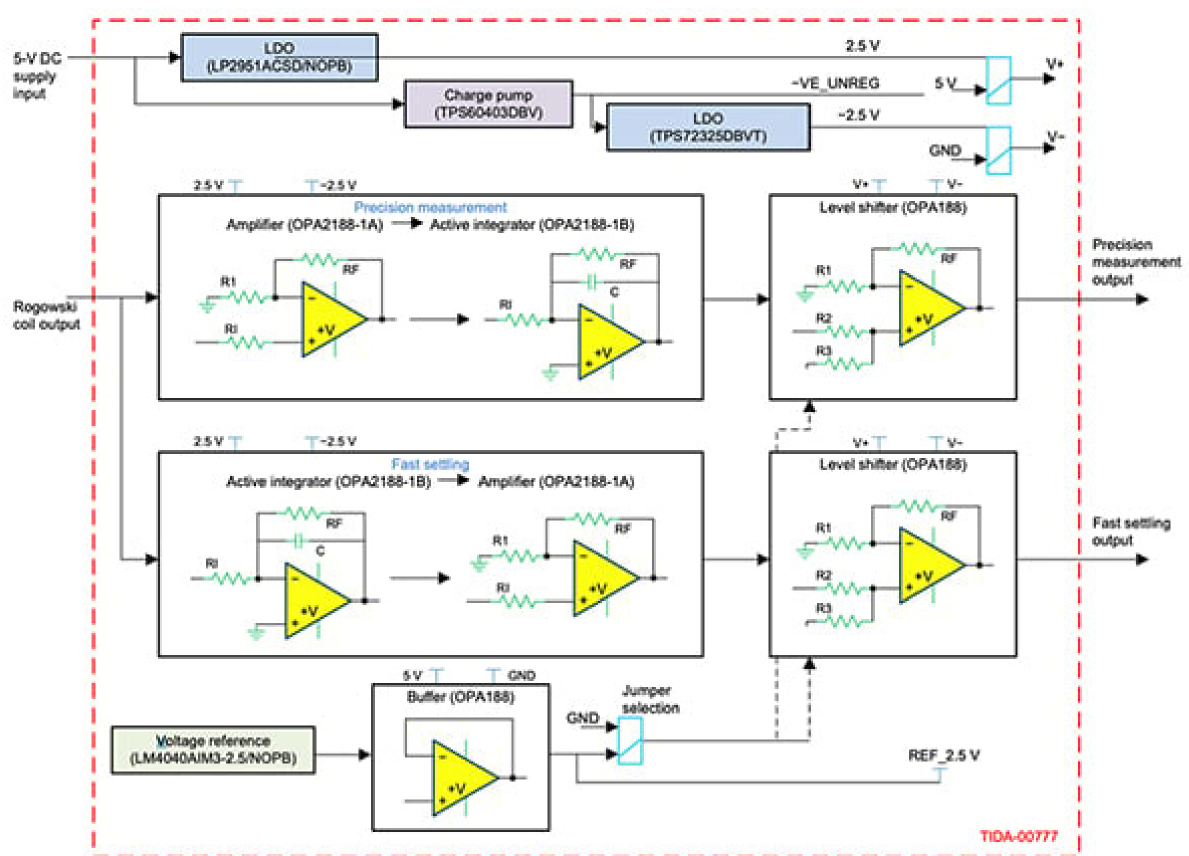


Figure 8: This reference design for a Rogowski coil integrator uses the Texas Instruments [OPA2188]() as the primary op amp in the integrator elements of the design. (Image source: Texas Instruments)

A reference design for a Rogowski coil integrator is shown in Figure 8. This design features both a high-precision output covering a range of 0.5 to 200 amps (A) with an accuracy of 0.5%, and a fast settling output over the same current range and an accuracy of within 1% in less than 15 milliseconds (ms).

The reference design uses Texas Instruments' OPA2188 as the primary op amp in the integrator elements of the design. The OPA2188 is a dual op amp that uses a proprietary auto-zeroing technique that results in a maximum offset voltage of 25 microvolts (μV) and near zero drift with time or temperature. It has a gain-bandwidth product of 2 MHz with an input bias current of ±160 pA, typical.

For this reference design, Texas Instruments selected the OPA2188 due to its low offset and low offset drift. Also, its low bias current minimizes the loading on the Rogowski coil.

Integrators in filters

Integrators are used in both state variable and bi-quad filter designs. These related filter types use dual integrators to obtain a second-order filter response. The state variable filter is the more interesting filter in that a single design yields simultaneous low pass, high pass, and bandpass responses. The filter uses two integrators along with an adder/subtractor stage, as shown in the TINA-TI simulation (Figure 9). The filter response for the low pass output is shown.

This filter topology has an advantage in that all three filter parameters—gain, cutoff frequency, and Q factor—are independently adjustable in the design process. In this example, the DC gain is 1.9 (5.6 dB), the cutoff frequency is 1 kHz, and the Q is 10.

Higher order filter designs are accomplished by placing multiple state variable filters in series. These filters are typically used for anti-aliasing in front of an analog-to-digital converter where high dynamic range and low noise are expected.

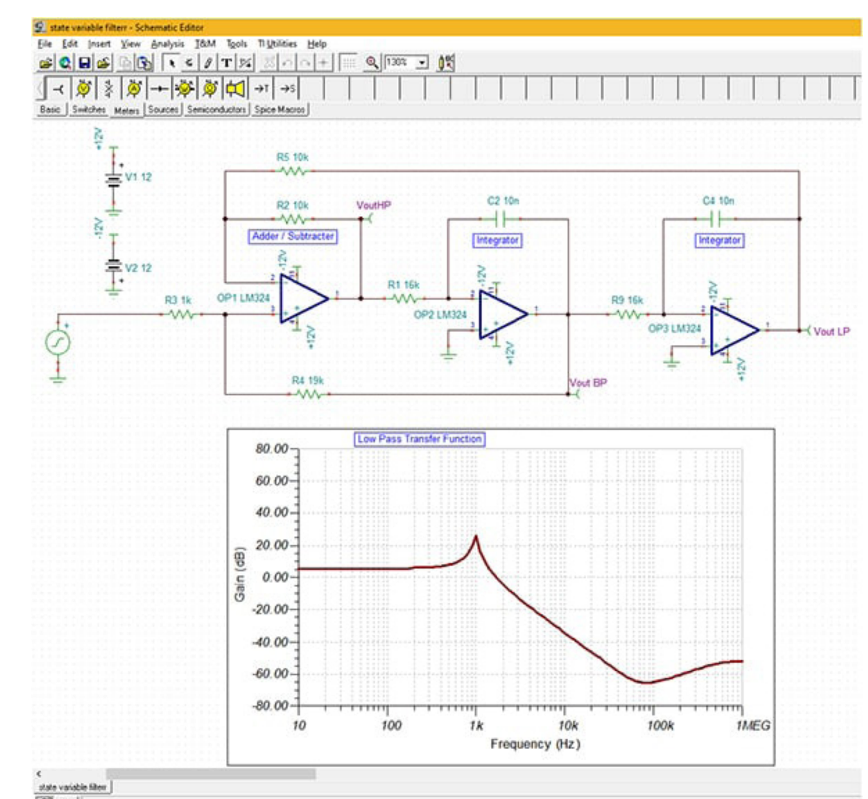
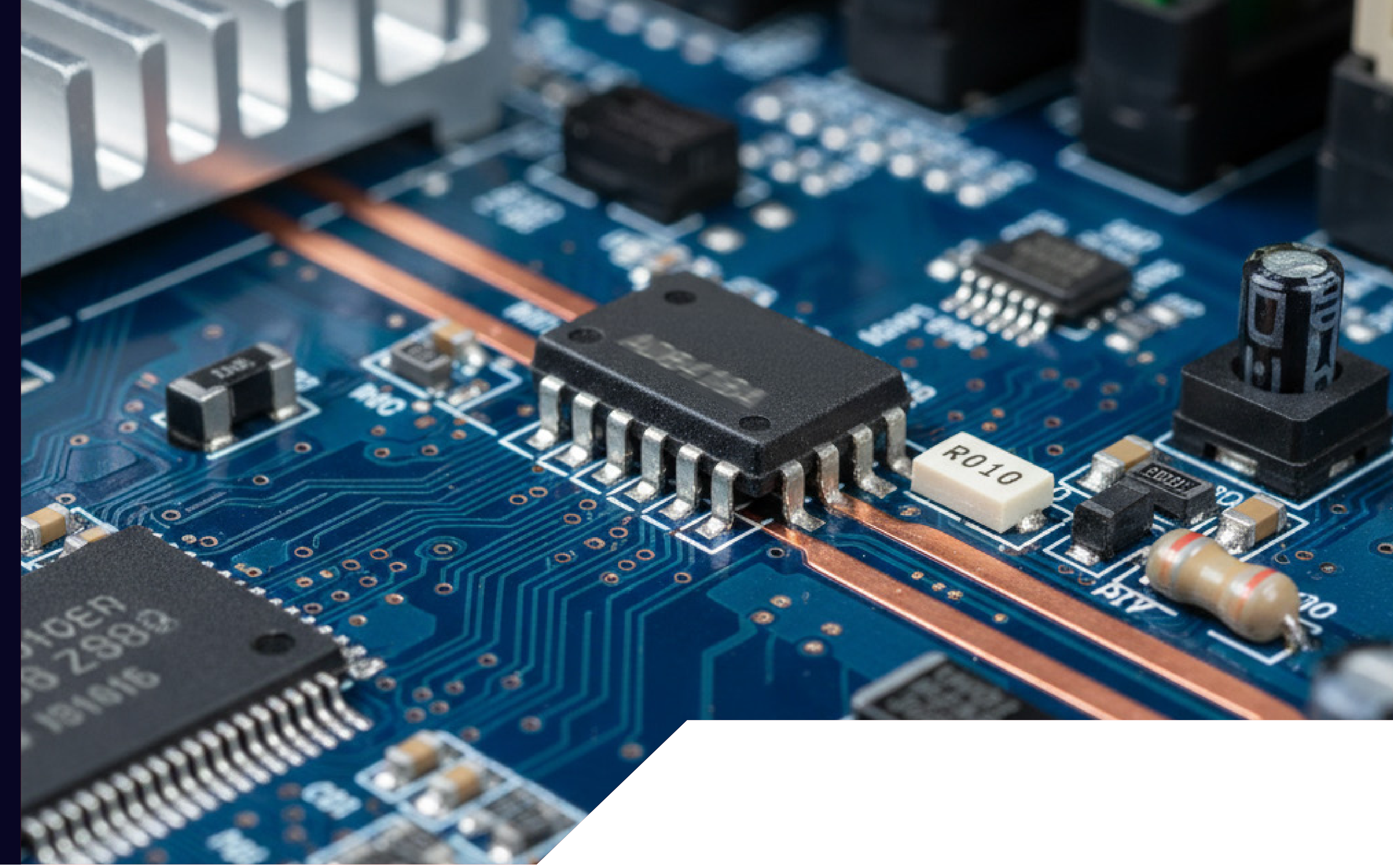


Figure 9: The state variable filter uses two integrators and an adder/subtractor stage to output low pass, high pass, and bandpass outputs from the same circuit. (Image source; DigiKey)

Conclusion

While it seems sometimes that the world has gone all digital, the examples discussed in this article show that the analog integrator remains an extremely useful and versatile circuit element for signal processing, sensor conditioning, signal generation, and filtering.

Efficiently implement current monitoring using integrated bidirectional current sense amplifiers



By Jeff Shepard
Contributed By DigiKey's North American Editors

Fast and accurate current monitoring is needed in a growing variety of applications including autonomous vehicles, factory automation and robotics, communications, server power management, Class-D audio amplifiers, and medical systems. In many of these applications, bidirectional current sensing is required, but it needs to be done efficiently and at minimal cost.

While it's possible to build a bidirectional current sense amplifier (CSA) using a pair of unidirectional CSAs, it can be a complex and time-consuming process. It involves a separate rail-to-rail op amp to combine the two outputs into a single-ended output, or the use of two analog-to-digital converter (ADC) inputs on the microcontroller, which requires additional microcontroller coding and machine cycles. Finally, building a bidirectional CSA using two unidirectional CSAs—plus the additional components required to integrate them into a bidirectional solution—can consume more circuit board space, and the higher parts count can reduce reliability and increase inventory requirements. The end result can be cost and design schedule overruns.

Instead, designers can turn to integrated, high-speed, precision bidirectional CSAs. They can select from integrated bidirectional CSAs with internal low-inductance shunt resistors that produce the most compact solutions, or CSAs that use external current shunts to provide a more flexible design and layout options.

This article reviews the implementation requirements of bidirectional CSAs and the benefits of a more integrated approach. It then introduces example devices from STMicroelectronics, Texas Instruments, and Analog Devices, including key parameters and differentiating characteristics. Finally, it shows how to get started on designs with these devices, including related reference designs/evaluation kits/dev kits, and tips on design and implementation.

How to use two unidirectional CSAs

A bidirectional CSA circuit can be constructed in more than one way using two unidirectional CSAs (Figure 1). The Analog Devices MAX4172ESA+T, used in the example on the left, does not include an internal load resistor, and so uses the discrete devices R_a and R_b. In the example on the right, the MAX4173TEUT+T has an internal 12 kilohm (kΩ) load resistor to convert its current output into a voltage.

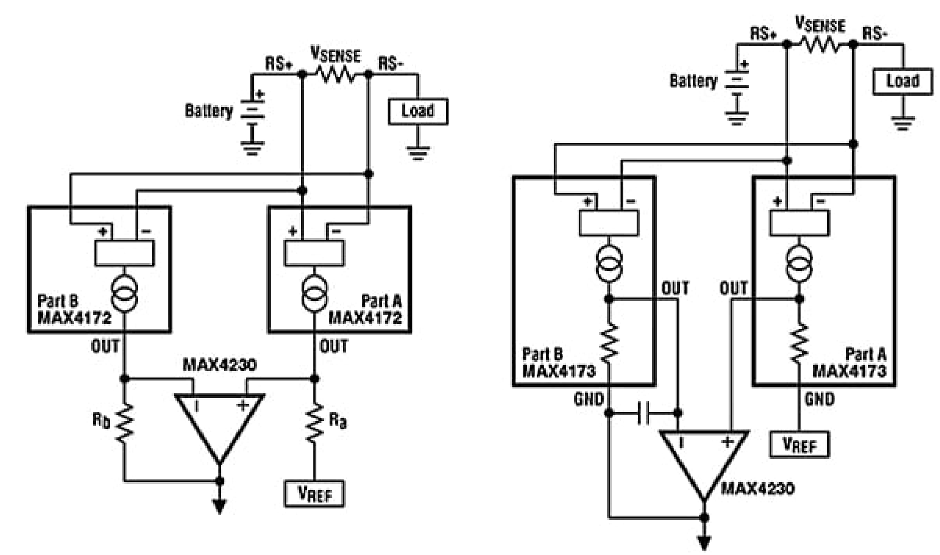


Figure 1: Bidirectional current sense applications using two unidirectional current sense amplifiers can be implemented using external load resistors (left), or with an internal load resistor (right). (Image source: Analog Devices)

While it does not need the two load resistors, the MAX4173TEUT+T circuit adds a 1 nanofarad (nF) capacitor in its

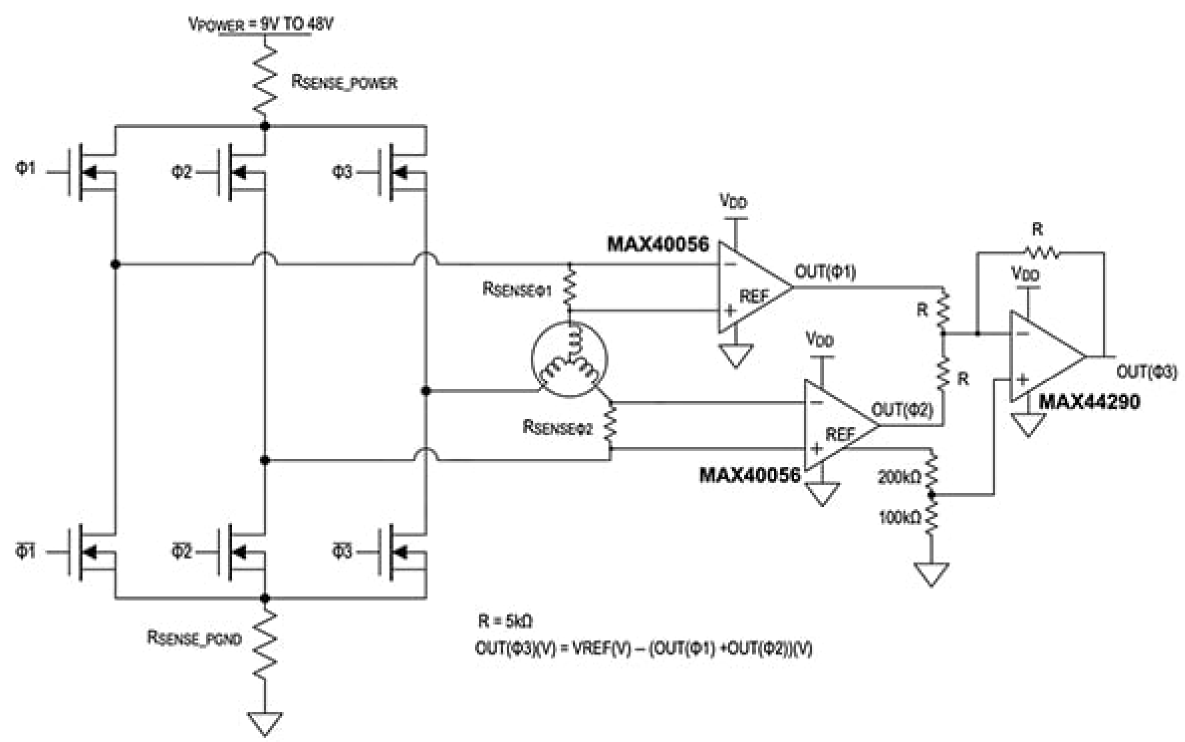


Figure 2: In a three-phase servo motor application, two bidirectional CSAs can be connected across sense resistors for phase 1 ($R_{SENSE\Phi1}$) and phase 2 ($R_{SENSE\Phi2}$) to generate a voltage representing the current in the third phase winding. (Image source: Analog Devices)

feedback to stabilize the control loop of Part B. In both cases, the output currents from the two CSAs are combined using a MAX4230AXK+T general purpose operational amplifier.

Both approaches have parts counts higher than what would be required using a single bidirectional CSA. In addition to the larger parts count, the pc board layout is more complex since both of the unidirectional CSAs need to be placed in close proximity to the V_{SENSE} resistor.

Application examples using bidirectional CSAs

Bidirectional CSAs are versatile devices and are found in a wide variety of applications. For example, two CSAs can be used in a three-phase servo motor system to determine the instantaneous winding currents of all three phases, without any further computation or information about the pulse width modulation (PWM) pulse phases or duty cycles (Figure 2).

Kirchhoff's Law states that the sum of the currents in the first two windings equals the current in the third winding. The circuit uses two MAX40056TAUA+ bidirectional CSAs to measure the two-phase currents which are summed in the MAX44290ANT+T general purpose operational amplifier. Since all three amplifiers have the same reference voltage, ratiometric measurements are produced.

In another example, a Class-D audio amplifier, a single bidirectional CSA such as the INA253A1IPW from Texas Instruments, can be used to accurately measure speaker load current (Figure 3).

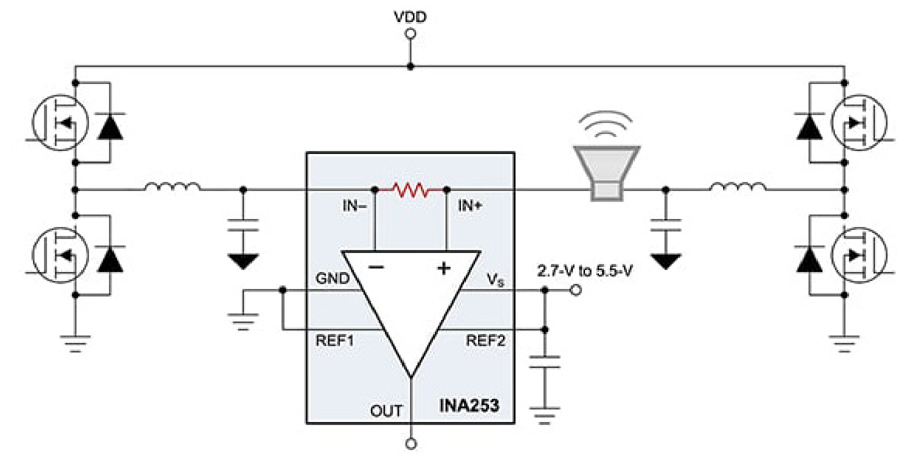


Figure 3: In Class-D audio designs, a bidirectional CSA (INA253) can be used to implement speaker enhancements and diagnostics. (Image source: Texas Instruments)

Real-time measurements of speaker load current can be used for diagnostics and to optimize amplifier performance by quantifying key speaker parameters and changes in those parameters including:

- Coil resistance
- Impedance of the speaker
- Resonant frequency and peak impedance at the resonant frequency
- Real-time ambient temperature of the speaker

Board layout tips and current shunt considerations

Parasitic resistance and inductance are a concern when implementing current sensing circuits. Also, excess solder and parasitic trace resistance can result in sensing errors. Four-terminal current sense resistors are often used. If a four-terminal resistor is not an option, the use of Kelvin pc board layout techniques should be followed (Figure 4).

Placing the Kelvin sense traces as close as possible to the current sense resistor's solder contact points minimizes parasitic resistances. A wider spacing of the Kelvin sense traces will introduce a measurement error caused by the additional trace resistance.

Sense resistor selection is an important aspect of minimizing parasitic inductance. Package inductances should be minimized since the voltage error is proportional to the load current. In general, wire-wound resistors have the highest inductance and standard metal film devices have mid-level inductances. For current sense applications, low-inductance metal film resistors are generally recommended.

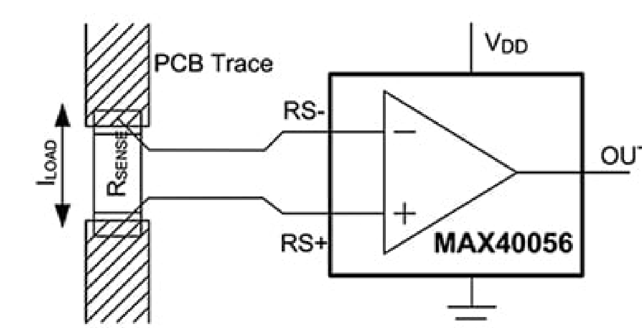


Figure 4: Kelvin sense traces should be as close as possible to the solder contact pads on the current sense resistor. (Image source: Analog Devices)

The value of the shunt resistor is a tradeoff between the dynamic range and power dissipation. For high current sensing, it is recommended that a low-value shunt be used to minimize the thermal dissipation (I^2R). In low current sensing, a higher resistance value can be used to minimize the impact of the offset voltage on the sensing circuit.

Most CSAs rely on external shunts to measure current, but there are some CSAs that use internal current shunts. While the use of internal shunts can result in more compact designs with fewer components, there are several tradeoffs involved including; less flexibility since the value of the shunt is predetermined, a need for a higher quiescent current compared with external shunt CSAs, and the amount of current that can be measured is limited by the capabilities of the internal shunt.

High voltage precision bidirectional CSAs

The TSC2011IST from STMicroelectronics enables designers to minimize power dissipation by taking advantage of its precision capabilities to use low resistance external current shunts (Figure 5). This bidirectional CSA is designed to deliver precision current measurements in applications such as data acquisition, motor

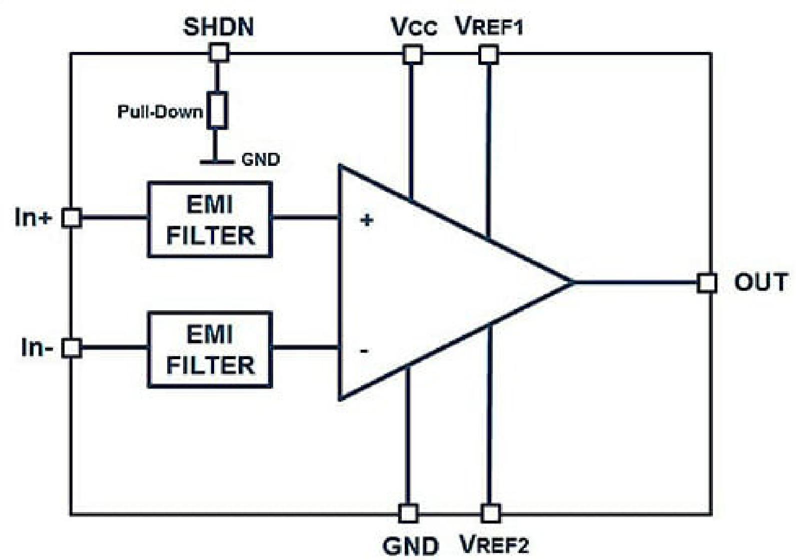


Figure 5: The TSC2011IST includes a shutdown pin (SHDN) to maximize energy savings, and it operates over the industrial temperature range of −40 to 125°C. (Image source: STMicroelectronics)

Figure 6: The STEVAL-AETKT1V2 eval board includes the main board and a daughter card containing the TSC2011IST. (Image source: STMicroelectronics)

control, solenoid control, instrumentation, test and measurement, and process control.

The TSC2011IST has an amplifier gain of 60 volts/volt (V/V), an integrated electromagnetic interference (EMI) filter, and 2 kilovolt (kV) human body model (HBM) electrostatic discharge (ESD) tolerance (according to JEDEC standard JESD22-A114F). The TSC2011 can detect a voltage drop as low as 10 millivolts (mV) full-scale to provide consistent measurements. Its gain-bandwidth product of 750 kilohertz (kHz) and slew rate of 7.0 volts per microsecond (V/µs) combine to ensure high accuracy and a fast response.

Designers can use the STEVAL-AETKT1V2 evaluation board to quickly get started using the TSC2011IST (Figure 6). It can sense current over a wide range of common mode voltages, ranging from -20 to +70 volts. The TSC2011IST features:

- Gain error: 0.3% max
- Offset drift: 5 µV/°C max
- Gain drift: 10 parts per million (ppm)/°C max
- Quiescent current: 20 microamperes (µA) in shutdown mode

Internal shunt bidirectional CSA

The INA253A1IPW from Texas Instruments integrates a 2 mΩ, 0.1% low-inductance current shunt and supports common mode voltages up to 80 volts (Figure

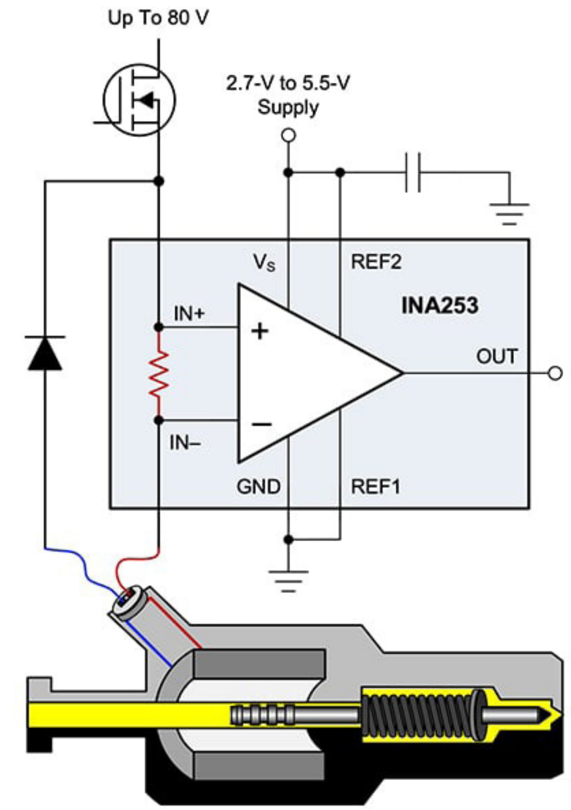


Figure 7: The INA253A1IPW bidirectional CSA, shown here in a typical application, has an internal current shunt and can measure ±15 A of continuous current from −40 to +85°C. (Image source: Texas Instruments)

7). The INA253A1IPW provides designers with enhanced PWM rejection circuitry to suppress large dv/dt signals, enabling real-time continuous current measurements for applications such as motor drive and solenoid valve control. The internal amplifier features a precision zero-drift topology with common-mode rejection ratios (CMRRs) of >120 decibels (dB) DC CMRR and 90 dB AC CMRR at 50 kHz.

Designers can accelerate the development of system designs based on the INA253A1IPW by using the test points on the associated INA253EVM evaluation board to access the CSA's functional pins (Figure 8). The two-layer board measures 2.4 × 4.2 inches and is fabricated with 1 ounce (oz) copper.

Minimal support circuitry is included on the pc board, and functions can be reconfigured, removed, or bypassed as needed. The INA253EVM provides the following features:

- Three INA253A1IPW devices
- Easy access to all pins
- Board layout and construction that supports ±15 A of current through the INA253 CSAs across the full −40 to +85°C temperature range
- Place holders on the pc board for configurations other than the default configuration

The bottom layer has no components but contains a solid copper ground plane that provides a low-impedance path for return currents.

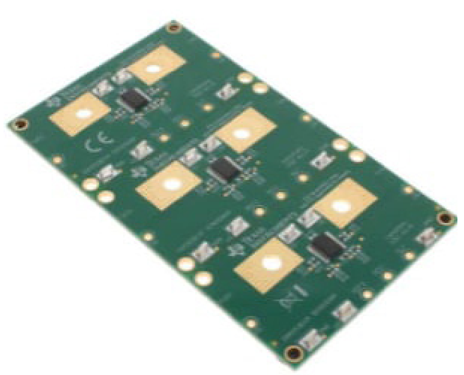


Figure 8: The two-layer INA253EVM measures 2.4 × 4.2 inches and is fabricated with 1 oz copper. The bottom layer has no components but contains a solid copper ground plane that provides a low-impedance path for return currents. (Image source: Texas Instruments)

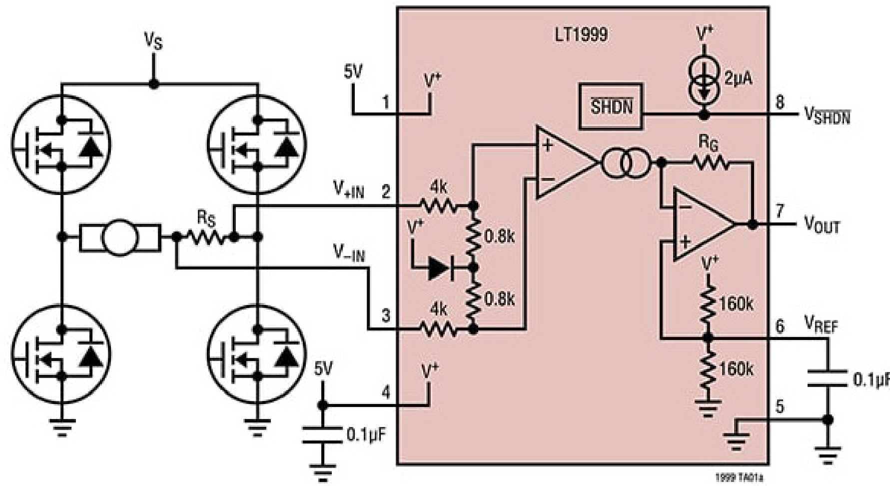


Figure 9: The LT1999IMS8-20#TRPBF is a bidirectional CSA in a full-bridge armature current monitoring application. (Image source: Analog Devices)

AEC-Q100 qualified bidirectional CSA

To monitor currents in full-bridge motor controls, switching power supplies, solenoids, and battery packs, as well as automotive applications, designers can use the LT1999IMS8-20#TRPBF from Analog Devices (Figure 9).

The LT1999IMS8-20#TRPBF is AEC-Q100 qualified for automotive applications and includes a shutdown mode to minimize power consumption. The device uses an external shunt to measure both the direction and amount of current flowing. It produces a proportional output voltage that is referenced midway between the supply voltage and ground. Designers have the option of applying an external voltage to set the reference level.

The LT1999IMS8-20#TRPBF enters a low-power shutdown state drawing about 3 µA when V_{SHDN} (Pin 8) is driven to within 0.5 volts of ground. The input pins (+IN and −IN) will draw approximately 1 nanoampere (nA) if biased within the range of 0 to 80 volts (with no differential voltage applied). EMI susceptibility is reduced by an internal 1st order, differential low-pass EMI suppression filter that helps reject high-frequency signals beyond the bandwidth of the device.

To experiment with the LT1999 series, Analog Devices provides the 1698A demonstration board. The board amplifies the voltage drop across an on-board current sense resistor and produces a bidirectional output voltage that is proportional to the current through the resistor. Designers can select from three fixed gain options; 10 V/V (DC1698A-A), 20 V/V (DC1698A-B) and 50 V/V (DC1698A-C).

Bidirectional CSA with PWM rejection

For improved rejection of common-mode input PWM edges in designs controlling inductive loads such as solenoids and motors, designers can use the MAX40056TAUA+ (Figure 10). Mentioned earlier in the context of Figure 2, the MAX40056TAUA+ is a bidirectional CSA that can handle slew rates of ±500 volts/µs and higher. It has a typical CMRR of 60 dB (50 volts, ±500 volts/µs input) and 140 dB DC. Its common-mode range is from -0.1 volts to +65 volts and includes protection against inductive kickback voltages down to -5 volts.

This MAX40056TAUA+ has an internal 1.5-volt reference that can be used for multiple purposes including:

- Driving a differential analog digital converter
- Offsetting the output to show the direction of the sensed current
- Sourcing current into external loads to mitigate performance reductions

When higher full-scale output swings are useful, or for supply voltages above 3.3 volts, designers can override the internal reference with a higher external voltage reference. Finally, designers can use either the internal or external reference to set the threshold for tripping the integrated overcurrent comparator, providing an immediate signal of an overcurrent fault.

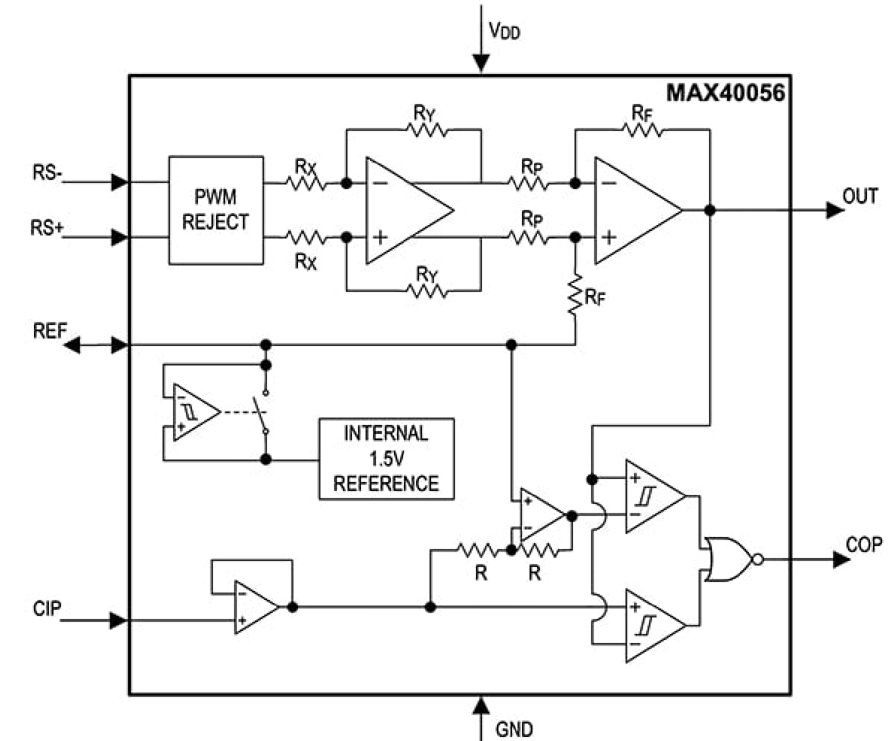


Figure 10: The MAX40056TAUA+ includes an internal 1.5-volt reference, enhanced PWM rejection, and an integrated internal window comparator to detect both positive and negative overcurrent conditions (bottom left, driven by the CIP input). (Image source: Analog Devices)

The MAX40056EVKIT# evaluation kit for the MAX40056TAUA+ provides designers with a proven platform for development of high-precision, high-voltage bidirectional CSA applications such as solenoid drives and servo motor controls.

Conclusion

Fast and accurate current monitoring is needed across a variety of applications, from automotive, factory automation, and robotics, to server power management, Class-D audio amplifiers, and medical systems.

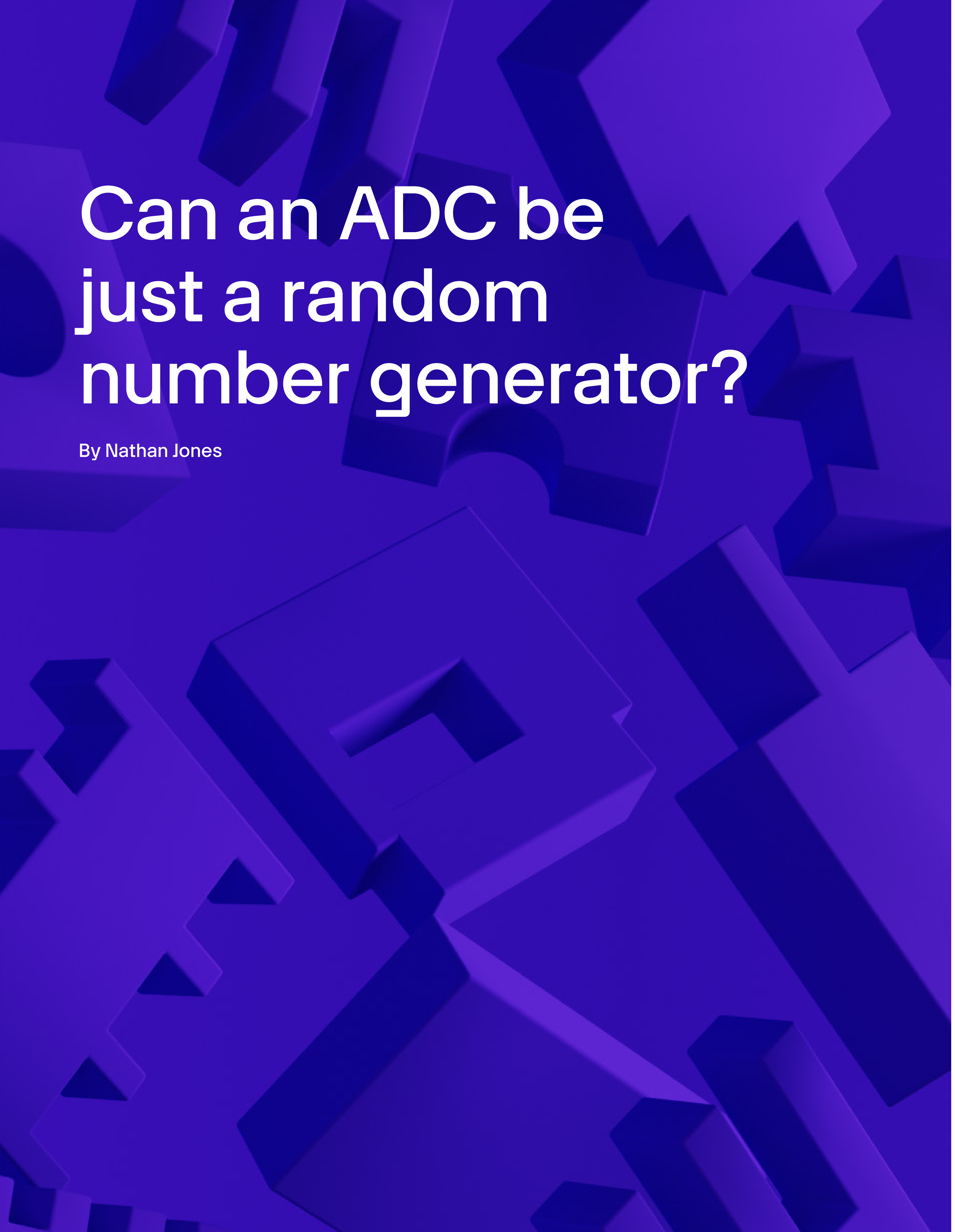
In many instances, bidirectional current sensing is needed.

Fortunately, designers can choose from a variety of integrated bidirectional CSAs, and their associated development platforms, to quickly and efficiently implement fast and accurate bidirectional current monitoring.

Recommended reading

[Use Sensorless Vector Control with BLDC and PMS Motors to Deliver Precise Motion Control](#)

[How to Choose and Use Angle Sensors for Power Steering, Motors and Robotics](#)



Can an ADC be just a random number generator?

By Nathan Jones

How good is an ADC? This question is not asking about how many bits of resolution it has; it may come as a surprise to know that a 16-bit ADC will not always give 16-bits of useful information! "A lot of 16-bit ADCs really operate as 7-bit ADCs and 9-bit random number generators," says Mike Stone in "Choosing an ADC". This article shows how to determine if an ADC is generating more random numbers than useful ones, along with a few simple ways to reduce the randomness.

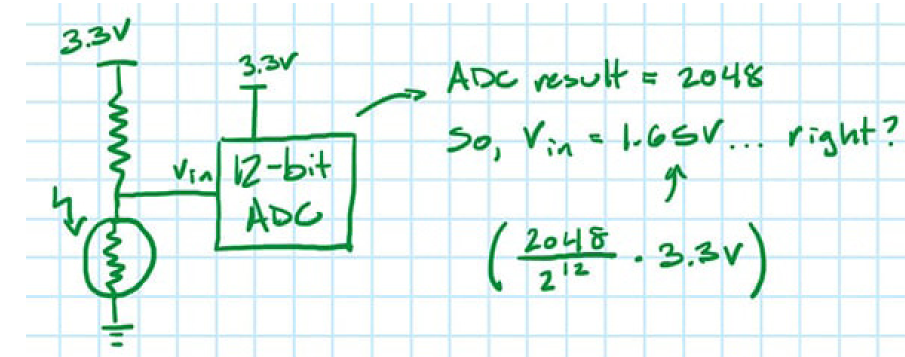


Figure 1: Simplified ADC setup for measuring a photoresistor voltage. (Image source: Nathan Jones)

Consider a simplified setup: an ADC measuring the voltage on a photoresistor. If the 12-bit ADC in Figure 1 (with V_{FS} = 3.3 V) reports a value of 2048, then does that mean the voltage on the photoresistor is 1.65 V?

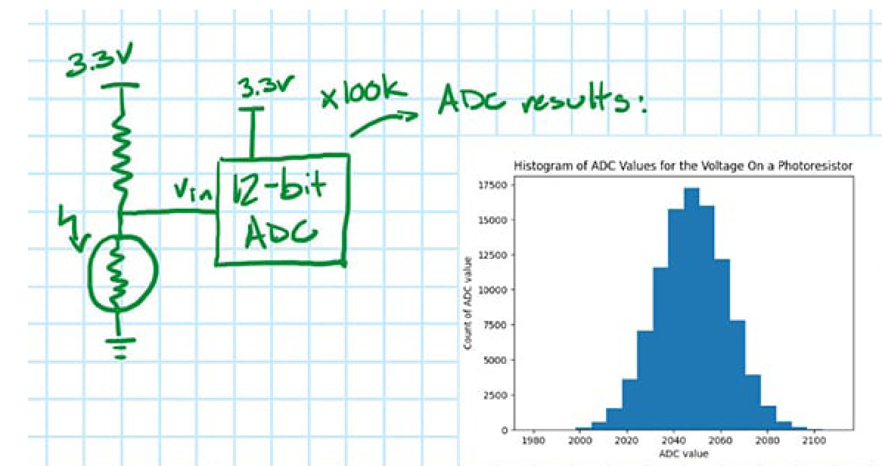


Figure 2: The insert shows the histogram of ADC values for 100,000 samples of photoresistor voltage. (Image source: Nathan Jones)

No, not really! To demonstrate, take a second sample, then a third, and, in fact, take 100,000 samples and plot the results. What will be seen is a *distribution* of ADC values (Figure 2); if nothing has changed in the setup, then any one of these could be the true voltage on the photoresistor, with the likeliest candidate being the average value of all those samples.

That brings up another question. Could some of that variation be *actual* variations in the input signal? Yes, it could. To test this, the resistive divider will be swapped out for a voltage reference (Figure 3) with a known noise level and the experiment repeated.

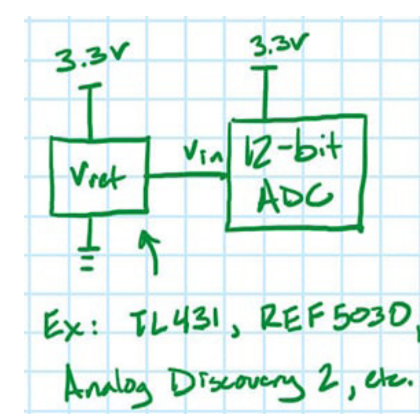


Figure 3: New test setup with a voltage reference in place of the resistive divider. (Image source: Nathan Jones)

Ideally the voltage reference used should have a noise level which is less than 1/3 of the noise level expected in the ADC (e.g., if the ADC is expected to have ±1 LSB of noise [which would be ±0.8 mV using the ADC in the example above], then ideally the voltage reference should have a noise level of less than 0.27 mV_{pp}); if it does, then it is expected that all of the measured noise must be coming from the ADC, based on the fact that uncorrelated noise adds in quadrature (Equation 1).

$$Noise_{total} = \sqrt{Noise_{ADC}^2 + Noise_{Vref}^2}$$

If $Noise_{Vref}$=$Noise_{ADC}$/3 then $Noise_{total}$=1.054 × $Noise_{ADC}$. This would mean that the voltage reference contributes just 5.4% of the total noise, a negligible amount. If it turns out that the voltage reference is noisier than 1/3 of the noise from the ADC, just remember to subtract its contribution from the measured standard deviation (see Equation 2).

$$Noise_{ADC} = \sqrt{Noise_{total}^2 - Noise_{Vref}^2}$$

How to know a voltage source's noise level

The easiest way is to find it on the datasheet. Here's an example from the datasheet for the Texas Instruments REF5030, showing that it has a noise level of 9 μV_{pp}.

If the datasheet doesn't list a noise level (or to just verify its value), just measure it with an oscilloscope, though the noise floor of the oscilloscope needs to be known first. This can be done by connecting one of the oscilloscope inputs to ground and measuring the RMS value of the variations in the signal seen. Figure 5 shows the result of doing this with an Analog Discovery 2 (AD2), demonstrating that the RMS noise is about 0.9 mV.

REF5030 (V_{OUT} = 3 V)						
V_{OUT}	Output voltage			3		V
	Initial accuracy, standard grade		–0.1%		0.1%	
	Output voltage noise	f = 0.1 Hz to 10 Hz		9		μV_{PP}

Figure 4: Excerpt from the REF5030 datasheet showing the output voltage noise specification. (Image source: Texas Instruments)

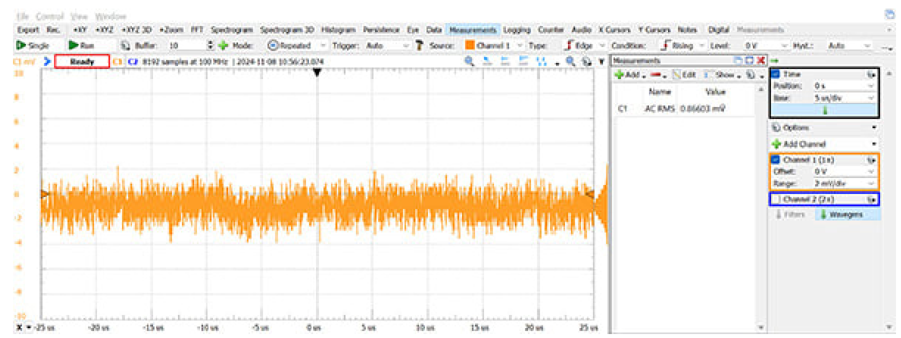


Figure 5: Noise floor level plot of the Analog Discovery 2 oscilloscope. (Image source: Nathan Jones)

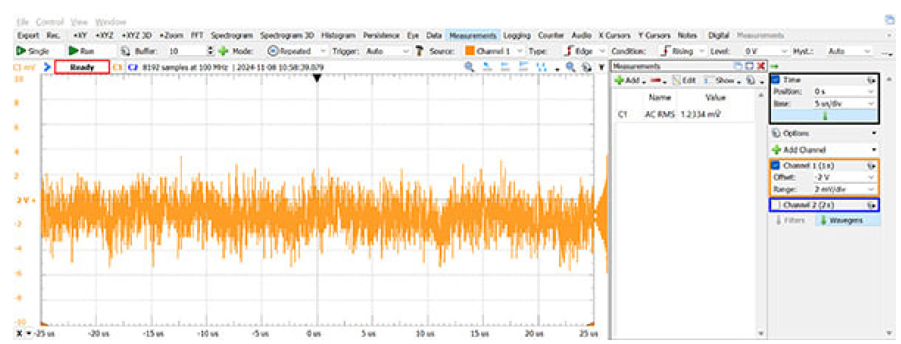


Figure 6: This plot of a 2 VDC signal indicates that the oscilloscope noise was 1.2 mV. (Image source: Nathan Jones)

Interesting, this value decreases (down into the µV range) if the time base is increased beyond 8 µs/div, presumably because the AD2 is using oversampling and decimation to increase its effective resolution.

When the voltage source is measured, any variations in the oscilloscope signal are the result of the combined noise in both the voltage source and the oscilloscope. As long as those noise sources are uncorrelated (they are), then $Noise_{measured}=\sqrt{(Noise^2_{Scope} + Noise^2_{Vref})}$ and $Noise_{Vref}=\sqrt{(Noise^2_{Measured} - Noise^2_{Scope})}$, like above. When a 2 V_{DC} signal that was generated by the AD2 was measured, the noise was determined to be 1.2 mV (Figure 6).

This would indicate that the signal generator on the AD2 also has a noise level of about 0.9 mV. This is low enough to test a 10-bit ADC (0.9 mV is just less than 1/3 of an LSB for a 10-bit ADC with 3.3 V full-scale reading); for any higher-resolution ADCs, this noise would have to be accounted for in the measurements.

That's much better! Notice now how the second histogram (Figure 7) only goes out to 2052 on the x-axis (the distribution on that graph [Figure 8] has a standard deviation of 1 LSB), while the first graph went out to 2100 (it had a standard deviation of 15 LSBs). This is called a **DC Histogram** test for an ADC and the standard deviation effectively indicates how much noise there is in the ADC: anytime a value is measured with the ADC, there's only a 68% chance that the *true* value on the analog pin is actually within ±1 standard deviation of that value, but there's a 99.7% chance that the true value is within ±3 standard deviations of that value.

Converting this standard deviation from LSBs to volts results in the **input-referred noise** for the ADC (Equation 3).

$$Input - referred\ noise\ [V_{RMS}] = \frac{\sigma \cdot V_{FS}}{2^N}.$$

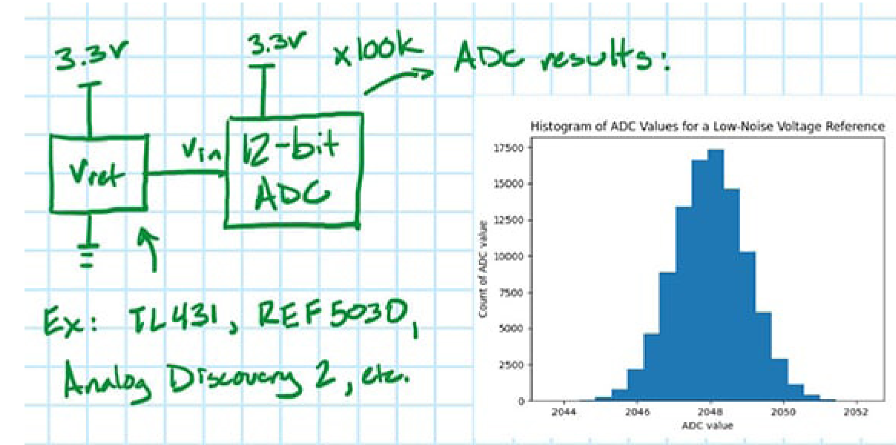


Figure 7: The insert shows the new histogram of ADC values for 100,000 samples of photoresistor voltage with the voltage reference. (Image source: Nathan Jones)

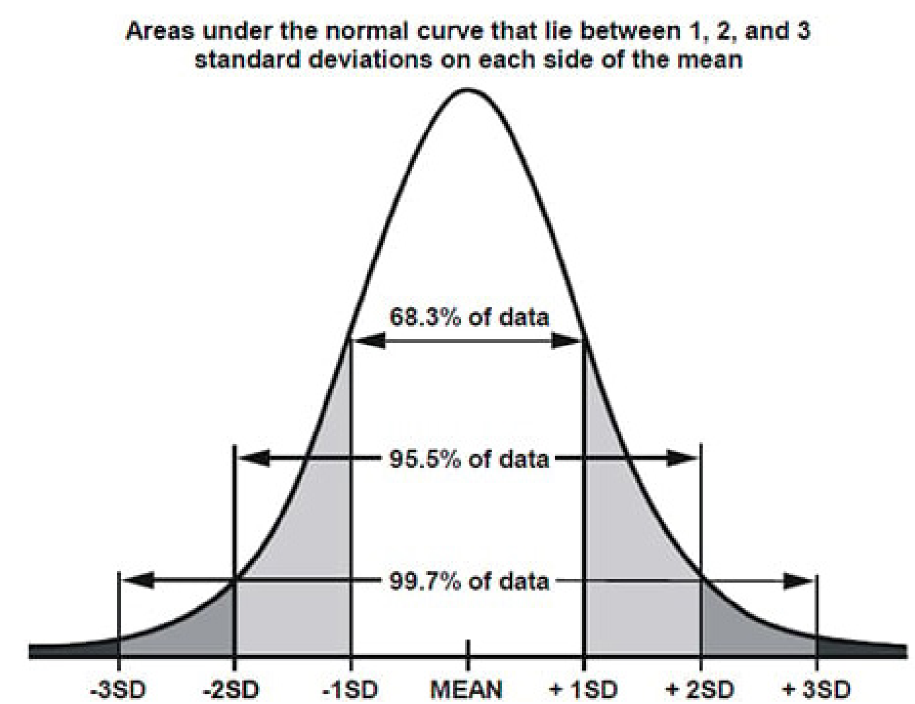
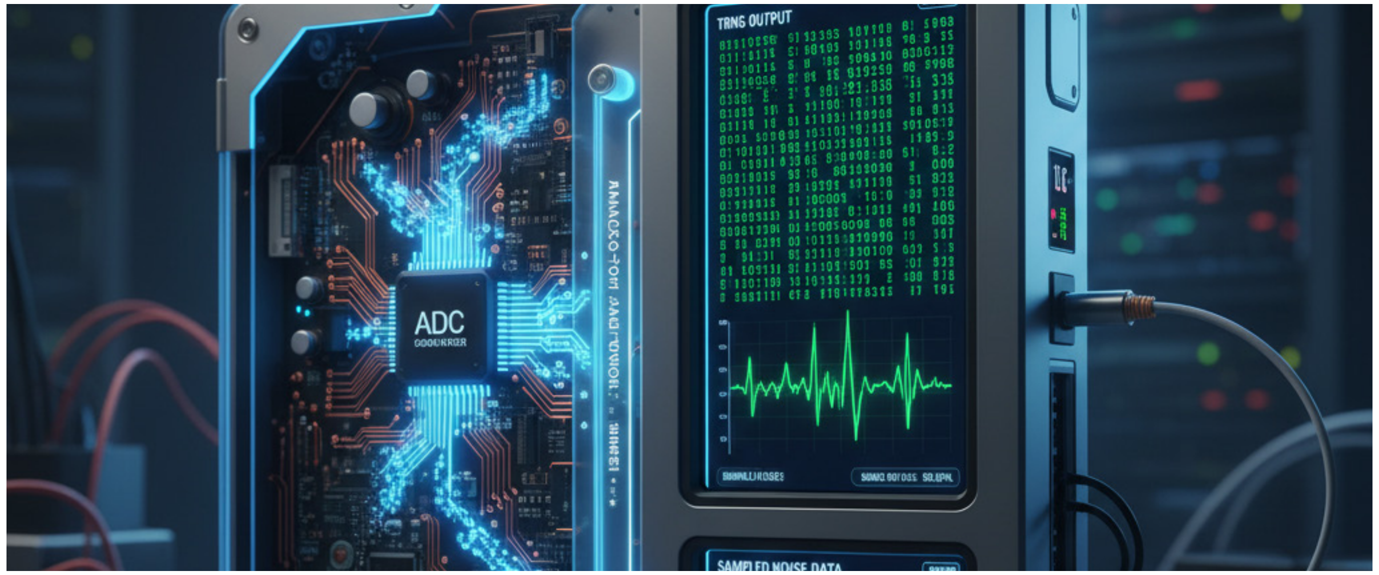


Figure 8: Distribution curve with standard of deviations. (Image source: https://i.sstatic.net/jkMDV.png)



If the ADC has such little noise that all ADC codes do fall into a single bin, try lowering the ADC reference voltage. The noise level will stay the same but will take up more ADC codes than before, allowing a proper value for the input-referred noise to be found.

The **effective resolution** and the **noise-free code resolution (NFCR)** of an ADC represent the extent to which this noise limits the ADC resolution, which are calculated by dividing the full range of ADC codes into bins that are only as wide as either one (Equation 4) or 6.6 standard deviations (Equation 5).

$$Effective\ resolution = \log_2 \frac{2^N}{\sigma}$$

$$Noise-free\ code\ resolution\ (NFCR) = \log_2 \frac{2^N}{6.6 \cdot \sigma}$$

Essentially, there are fewer actual bits in the result because there's an increasing chance that the last few bits in the ADC result are wrong or noisy.

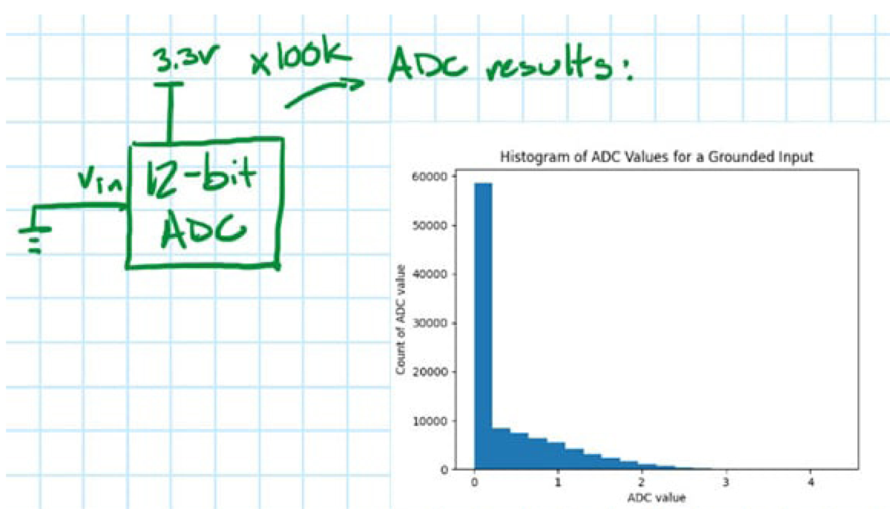


Figure 9: The insert shows the histogram of the ADC output voltage samples with the input grounded. (Image source: Nathan Jones)

A slightly more straightforward version of the DC Histogram test involves simply grounding the input of the ADC and then creating a histogram (Figure 9) of the ADC results, as was done for the oscilloscope.

The standard deviation of this histogram is *also* a measure of the ADC noise (minus any contributions from noise in the ADC reference voltage, which won't affect readings of 0 V), but it's optimistically low. The ADC probably reports values less than 0 V as just 0, so only *half* of the true noise distribution is being seen. In this case, the ADC noise could be approximated if it is assumed that the distribution was symmetric about its average value, changing enough of the median values to negative numbers so that the distribution became symmetric and then calculating the standard deviation of *that* (Figure 10).

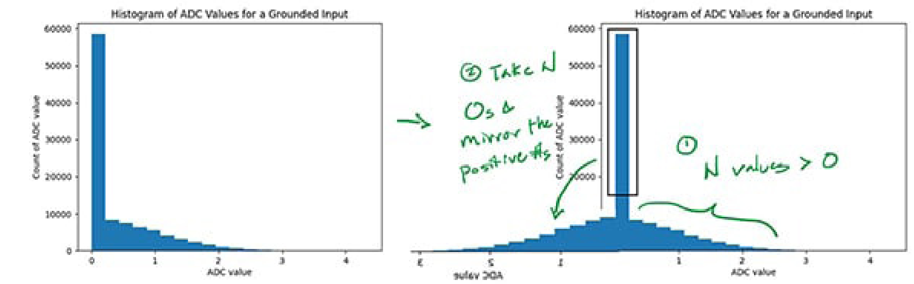


Figure 10: Mirroring of the Figure 9 histogram to approximate the ADC noise. (Image source: Nathan Jones)

Okay, so what can be done about all this noise? If the resolution of the ADC is still within the project requirements (lucky!), then just keep in mind that every ADC measurement comes with some uncertainty:

- When comparing two ADC values, treat them as being equivalent only if they're within 6.6 σ of each other (to be 99.7% confident in the comparison) or if they're within σ of each other (to be only 68% confident).
- When comparing an ADC value to a threshold, add some hysteresis to the threshold comparison (like a thermostat does) to avoid the embedded system from re-triggering above and below that threshold just based on noise.

If it turns out that the ADC doesn't meet the project requirements anymore, there are a few simple things to try to improve its performance. On the hardware side, these include:

- Using a lower-noise reference (such as the Texas Instruments REF5030 or the EVVO TL431) for AVCC and AREF
- Ensuring minimal crosstalk between analog and digital signals on the PCB
- Turning off the CPU during ADC conversions (if the MCU supports it), to reduce digital noise

On the software side, the input signal can be oversampled (i.e., take the average of N samples) to reduce noise, though this will clearly reduce the maximum sampling rate. (In fact, a technique called "oversampling and decimation" could be used to increase the resolution of the ADC by an amount related to the number of times the input is oversampled!)

Assume the standard deviation of a DC Histogram test for the initial setup is 1 LSB. If the ADC reads 2048, *now* does that mean (there's a 68% chance) the true value is 1.650±0.008 V (Figure 11)?

Closer, but still no! That's because all ADCs will struggle with measuring a rapidly changing input signal at some frequency, resulting in imprecise measurements (i.e., noise).

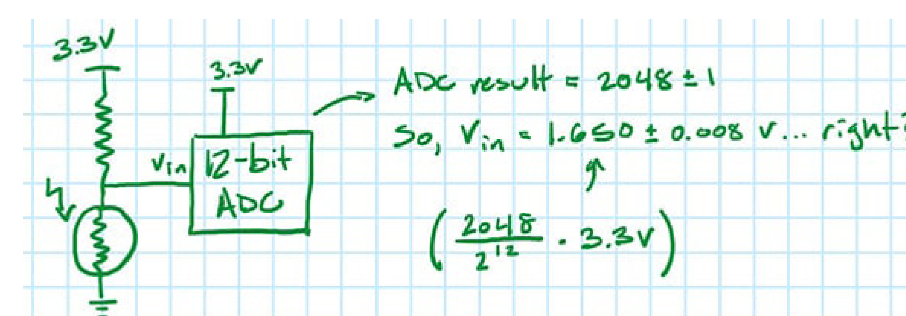


Figure 11: Assuming the standard of deviation of the Figure 1 setup is 1 LSB and the ADC output is 2048, is the true input voltage 1.650±0.008 V? (Image source: Nathan Jones)

Surprisingly, the frequency at which that occurs may be well below the maximum sampling rate for the ADC! This means that a 12-bit ADC with a sampling rate of 1 MSPS (mega-samples per second) may not actually have 12-bits of resolution for input signals near 500 kHz; the resolution of that ADC may fall to 6-bits or lower when the input signal frequency is that high. If trying to measure a signal on the photoresistor that's above 10 Hz, the ADC may not actually have 12-bits of resolution.

To determine how well the resolution of the ADC holds up with higher input frequencies, send in a known-good signal (this time, a sine wave) and measure how closely the ADC values match that known-good signal (Figure 12). (Is the sine wave "good enough"? That will be discussed after the procedure for performing the test is revealed; it will make more sense that way.)

For the sine wave, a function generator can be used (like the ones found on many USB oscilloscopes), a DDS chip (like Analog Devices' AD9834), or a custom sine wave generator can be assembled (based on a Wien bridge oscillator or by using a MAX7400 from Analog Devices to lowpass filter a square wave, for example). It is important to note that only input frequencies for this test that are not integer sub-multiples of the sampling frequency should ever be used (e.g., don't use an input frequency of 1 kHz if the sample rate is 100 kHz). This is to avoid erroneously sampling the sine wave in the same locations on every cycle. Additionally, to maximally stress the ADC, set the input sine wave's amplitude to the largest value that doesn't "clip" the output (e.g., something slightly less than 1.65 V for the ADC in the examples above).

After collecting the ADC results, the data is fit to a sine wave to calculate the residual error. The sine wave fit can be performed with almost any amount of data, but to get the best results, there has to be at least five full periods of the input sine wave. It is likely that the sine-fitting algorithm will have to be initialized by giving it initial guesses for the amplitude, frequency, phase shift, and offset (Figure 13).

This test is called the **sine wave fit test** for an ADC, and the RMS value of the residual error is a measure of how noisy the ADC is at that specific input frequency. At lower frequencies, the RMS noise should match the input-referred noise from the DC histogram test, but at higher frequencies it will almost certainly degrade as a result of those frequency-dependent noise sources mentioned above.

The RMS noise value from the sine wave fit test can be used to calculate the **effective number of bits (ENOB)** of the ADC (Equation 6), which is the number of bits of an ideal ADC whose only noise source, quantization noise, has the same RMS value as the noise in the ADC.

$$Effective\ number\ of\ bits\ (ENOB) = \log_2 \frac{2^N}{\sqrt{12} \cdot Error_{RMS}}$$

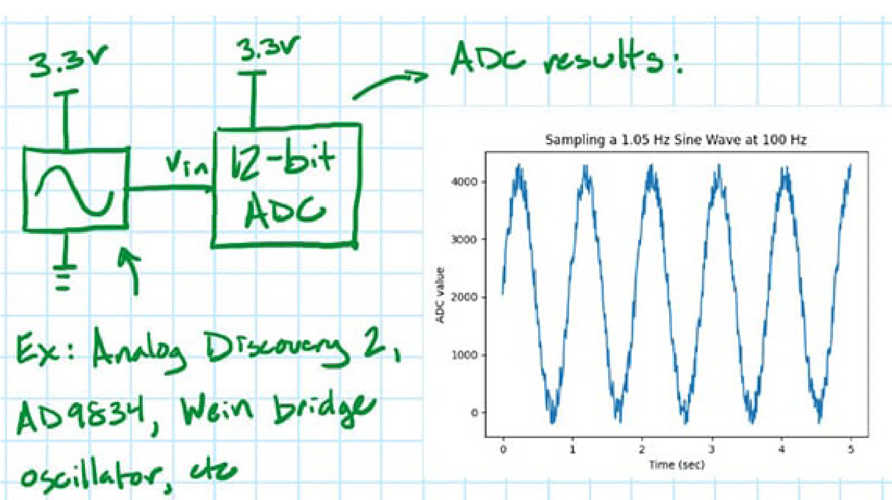


Figure 12: Test setup with a known-good sine wave input. (Image source: Nathan Jones)

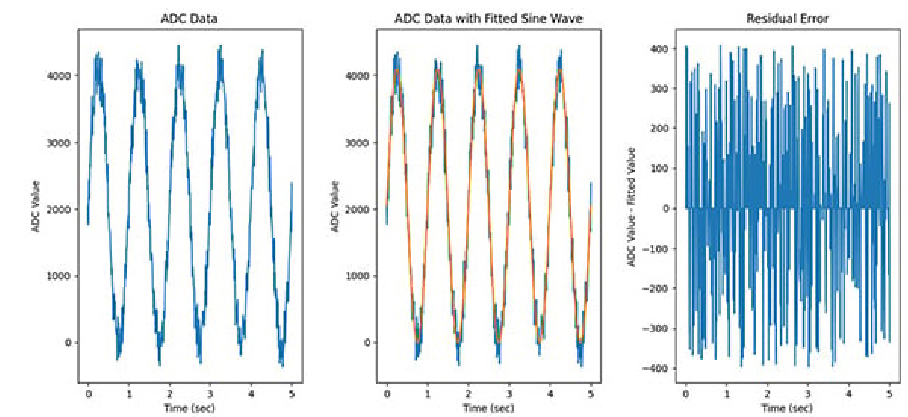


Figure 13: Using a sine wave fitting algorithm, the residual error can be determined. (Image source: Nathan Jones)

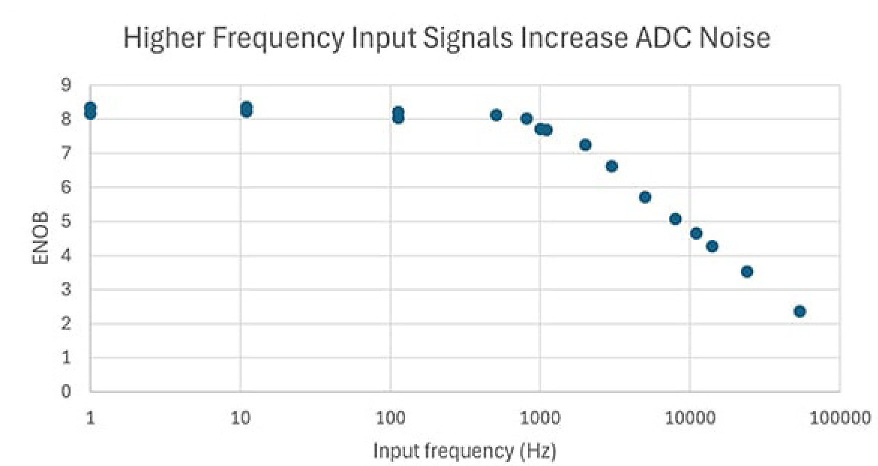


Figure 14: Frequency noise test for an ADC in an STM32F042K6 microcontroller. (Image source: Nathan Jones)

This value is specific to the frequency of the input signal that was used. To get a more complete picture of the ADC, this test should be repeated at frequencies all the way up to the highest input signal of interest, or up to half of the maximum sampling rate. It may be found that the noise goes way up (and the ENOB way down!) at frequencies well below half of the maximum sampling rate! For example, that exact series of tests were performed on the ADC in an STMicroelectronics STM32F042K6 and plotted the results (Figure 14).

Even though the ADC on the STM32F042K6 microcontroller can sample at 1 MHz, the resolution falls off above 1 kHz; if the intent was to measure a 10 kHz signal with this ADC, there would only be about 5 bits of resolution at that point! (Notice, also, that the maximum ENOB is around 8.3; this is due to ~10 LSBs of noise even at DC, which reduce the effective resolution of the ADC from 12 bits to around 8.7 bits right off the bat. These tests were conducted on an unmodified STMicroelectronics Nucleo-F042 development board; results would be much better if any of the techniques mentioned above are used ["Okay, so what can be done about all this noise?"].)

How to know a sine wave's noise level

As with the voltage reference, if the noise level of the sine wave is a *priori* (i.e., if it's not available on a datasheet), then there is a need to measure it with an oscilloscope. The use of the sine wave fitting technique discussed earlier can characterize the noise from the sine wave generator.

1. First, determine the noise floor of the oscilloscope by connecting its input to ground and calculating the RMS value of the output (as described earlier in the "How to *know* a voltage source's noise level" section).

2. Then, measure the output from the sine wave generator, fit the results to a perfect sine wave, and calculate the RMS value of the residuals. Then use Equation 2 to determine the noise of the sine wave generator (using the oscilloscope's noise floor as an approximation of its AC noise).

Ideally, the noise level of the sine wave is less than 1/3 the noise level expected from the ADC (in other words, the sine wave is "spectrally pure"), which would make its contribution to the measured noise negligible. If the noise level of the sine wave is higher than that, remember to subtract it from the measured noise obtained from calculating the RMS values of the residual error.

This test should be repeated for every input frequency that is intended to be used to test the ADC (even the sine wave generator may have frequency-dependent noise sources!).

What can be done about all this noise? First, it is recommended to put a low-pass filter (LPF) in between the signal source and the ADC, with a cutoff frequency just above the highest signal of interest, to help limit high frequency noise (this is a "good thing to do", in general, for any data acquisition system). If that highest frequency signal of interest is below the "knee" in the graph of ENOB vs Input frequency, then there really isn't any loss of resolution. When trying to measure signals above that "knee", there are a few things to try to improve that resolution at higher frequencies. These include:

- Ensuring that the output impedance of the device that's connected to the analog pin is very low, possibly inserting an op-amp for buffering
- Adjusting the ADC clock or its sample/hold time to be as fast as possible, while still giving the internal capacitor enough time to charge
- Using a low-jitter ADC clock source and configuring the system to take ADC samples at regular intervals

Unless a signal in a very narrow band of frequencies is the goal, the most straightforward thing to do next is to assume that the actual input signals have equal frequency content from DC up to the cutoff frequency of the LPF, and therefore that the RMS noise in every ADC measurement is the average RMS noise measured using the sine wave fit test from DC up to the cutoff frequency. If it is desired to measure broadband signals up to 10 kHz using the STM32F042K6 discussed earlier, then it would be best to operate as though the ADC only had an ENOB of around 6 bits for each measurement. This value may very well be lower than the one determined using the DC Histogram test, indicating that there is less certainty in each of the ADC measurements if there's a possibility that they have higher frequency content.

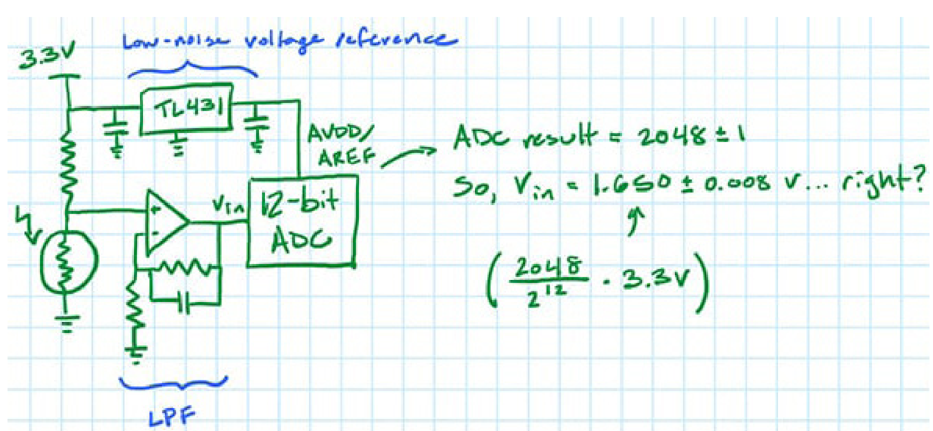


Figure 15: Does the addition of a low-pass filter in the circuit result in a 1.650±0.008 V true value? (Image source: Nathan Jones)

The next assumption to consider is the addition of an LPF in between the sensor and the ADC such that the RMS noise from DC up to the cutoff frequency is only ±1 LSB. If the ADC reads 2048, now does that mean (there's a 68% chance) the true value is 1.650 ±0.008 V (Figure 15)?

As crazy as it sounds, *still not yet!* Although this article has effectively characterized the *noise* in the ADC, it hasn't yet characterized its *error*. After the testing outlined here, the only confidence is that two ADC values are the same or different. However, it is unknown as to whether an ADC code of 2048 corresponds to *exactly* 1.65 V or to something else. Gain, offset, and differential non-linearity errors (among others) could mean that converting from an ADC code to an actual voltage is more complicated than using V_{in}=*ADC result* × $V_{FS}/2^N$. But that's a topic for another article!

References

1. Characterizing the Raspberry Pi Pico ADC
2. Understanding Data Converters (Texas Instruments)
3. Understand SINAD, ENOB, SNR, THD, THD + N, and SFDR so You Don't Get Lost in the Noise Floor (Analog Devices)
4. The Good, the Bad, and the Ugly Aspects of ADC Input Noise--Is No Noise Good Noise? (Analog Devices)
5. Getting the most out of the SAM D21's ADC (Thea Codes)
6. "Analog-to-Digital Converter Testing" (Kent Lundberg)
7. Dynamic Tests For A/D Converter Performance (Texas Instruments)
8. How to optimize the ADC accuracy in the STM32 MCUs (STM)
9. Enhancing ADC resolution by oversampling (Atmel)
10. Fundamentals of Precision ADC Noise Analysis (Texas Instruments)
11. ADC Gain and Offset Error Calibration on ARM® Cortex®-M0+ Based MCUs (Microchip)
12. Sine wave generation via low-pass filtering a square wave
13. Understanding the impact of digitizer noise on oscilloscope measurements (EE Times)
14. Understanding Effective Number of Bits (Robust Circuit Design)

Video spotlight

TSZ High Precision Zero Drift OpAmps

The TSZ series features ultra-high precision, low-power, zero-drift operational amplifiers in compact, space-saving packages, tailored for automotive and industrial applications. Built on a chopper-stabilized architecture, they deliver exceptionally low offset voltages and virtually zero drift for superior performance.

Learn more

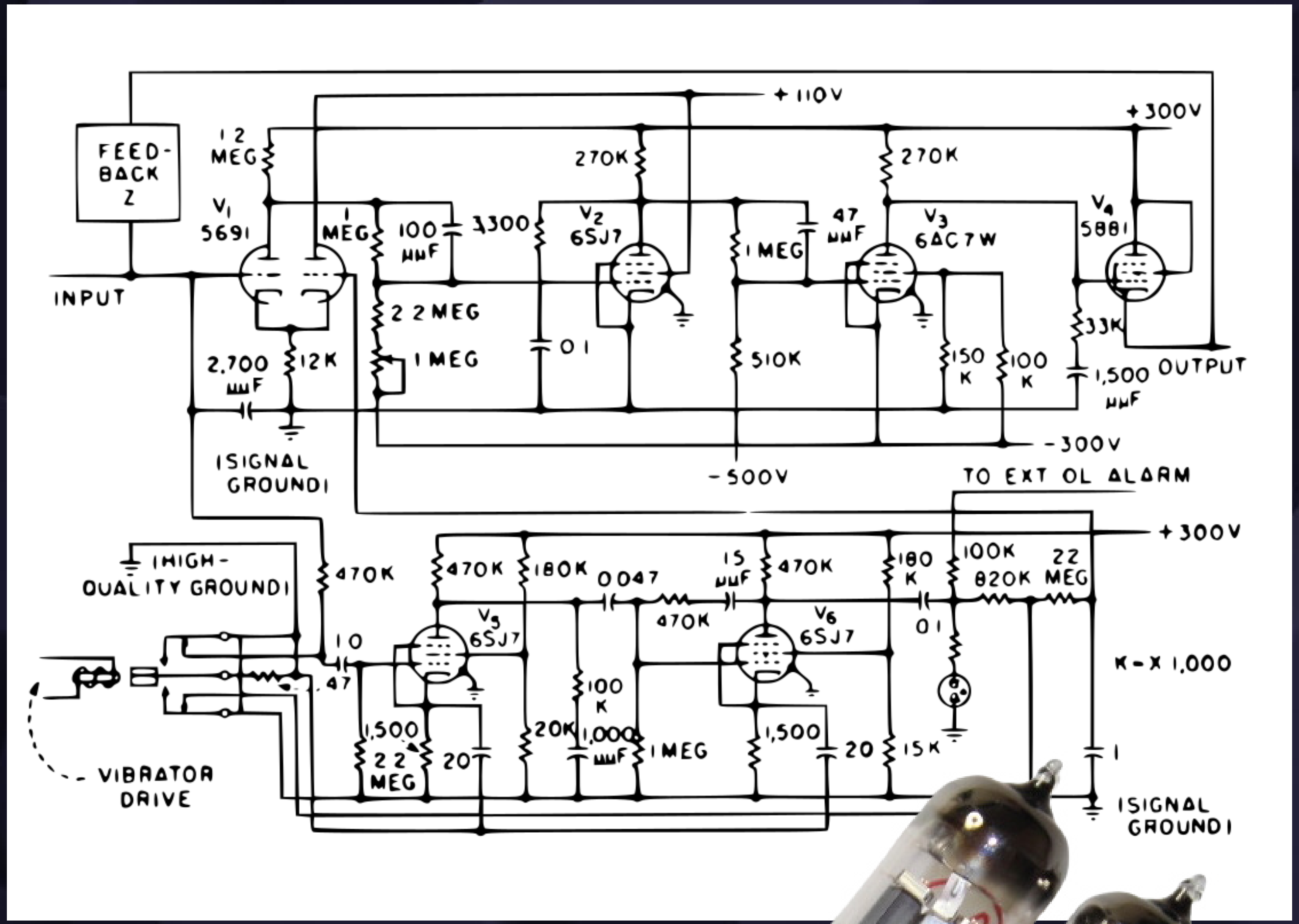
8-Channel Dual Simultaneous-Sampling ADC

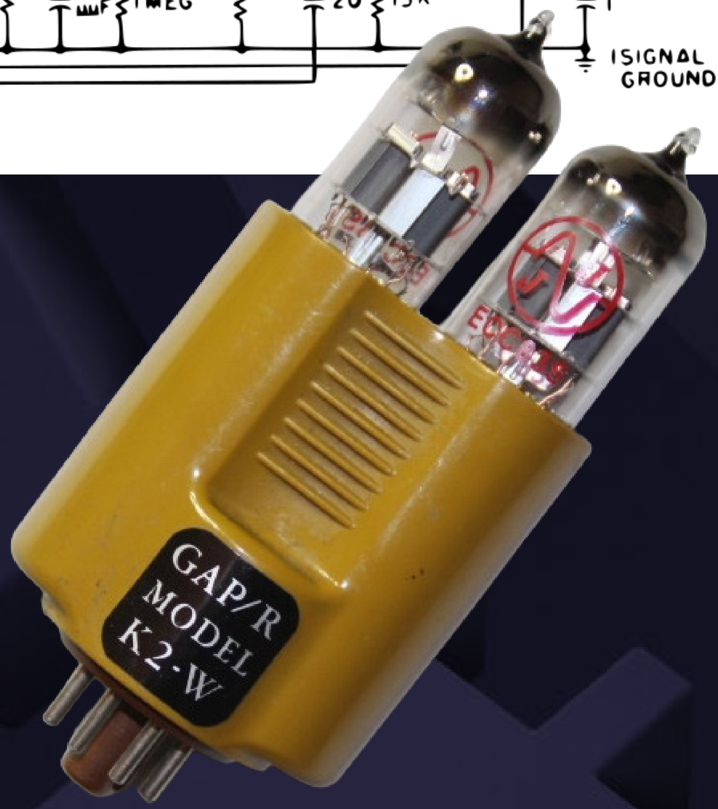
Texas Instruments' ADS981x is an 8-channel, dual simultaneous-sampling analog-to-digital converter (ADC) featuring an 18-bit successive approximation register (SAR) ADC with up to 2 MSPS per channel. This Product Information Overview video explains the features and benefits of the ADS981x.

Learn more

Teaching an amplifier new tricks: The story of the operational amplifier

By David Ray, Cyber City Circuits





Should an amplifier be trusted at all?

An operational amplifier, or op-amp, is not defined by what it contains or how it is built, but by how it is used. At its core, an op amp is a high-gain electronic device with two inputs and one output, designed so that its internal behavior is deliberately overwhelmed by external control. Left alone, it is unstable and nearly useless, so sensitive that a fraction of a millivolt will drive it hard into saturation. Only when wrapped in feedback does it become precise, predictable, and extraordinarily useful.

What makes the op amp special is not the act of amplification itself, as engineers have been amplifying signals for decades, but rather that the amplifier's job is no longer to determine its own gain. Instead, gain, bandwidth, linearity, and even function are decided from the outside by a small network of passive components. The active device becomes an obedient intermediary, enforcing physical laws rather than asserting its own personality.

Once constrained in this way, the op amp can be made to add, subtract, integrate, differentiate, filter, buffer, or compare signals with remarkable accuracy. The same basic device can serve as a sensor front-end, an audio amplifier, a control-loop element, or the mathematical heart of an analog computer.

Before the operational amplifier could exist, engineers first had to answer a more fundamental question: ***Should an amplifier be trusted at all?***

Harold S. Black: designer of the negative feedback amplifier

This was the problem **Harold S. Black** confronted in the 1920s while working on long-distance telephone systems. Transcontinental telephony depended on chains of vacuum-tube repeaters spaced along thousands of miles of copper wire. Each amplifier restored lost signal strength, but also added its own distortion, noise, and drift. As more repeaters were added, those imperfections compounded until speech became hollow, unstable, and often unintelligible.

The industry's instinct was to pursue better devices: more linear tubes, tighter tolerances, constant adjustment. Black took a different approach. Instead of trying to perfect the amplifier, he asked whether its imperfections could be made irrelevant. His answer, negative feedback, forced an amplifier to compare its output to its input and correct its own errors continuously, without constant adjustments. The gain was reduced, but control was preserved, distortion was suppressed, and stability was gained, resulting in a working transcontinental telephone network.

Harold S. Black

Black's feedback amplifier transformed amplification from a fragile craft into a controllable system. Crucially, it shifted precision away from the active device and into the surrounding network. An amplifier no longer needed to be accurate, it needed only to be powerful enough to obey the constraints placed upon it.

That single concept solved the telephone problem, but it also unlocked something new. Once an amplifier could be trusted to behave predictably under feedback, engineers were free to ask a new question: *If an amplifier can be tamed, what else can it be made to do?*

Retro Electro fun fact: Harold Black spent years working on this problem at Western Electric (later Bell Laboratories). For several years, the patent office wouldn't approve the Negative Feedback Amplifier because they thought it was impossible and could never work. Read all about the story of the origins of Bell Labs and the first negative feedback amplifier in the Retro Electro article: **Engineering Silence**. (www.digikey.com/en/emedia/emagazine/2025/robotics?page=11)

Teaching tricks to an amplifier

The first operational amplifier is considered to be "The Summing Amplifier" by **Karl D. Swartzel Jr.** at Bell Labs. Born in 1907 in Ohio, K. D. Swartzel Jr. was a gifted student. His father, Karl D. Swartzel, was an established mathematics teacher and trustee at Ohio State University. Concepts like abstractions, balance, and constraints would have been very familiar to him long before he met a vacuum tube.

Later, in 1922, his father moved with his family to Pittsburgh to serve as head of the mathematics department at the University of Pittsburgh. Swartzel Jr. graduated from the University of Pittsburgh with a degree in mathematics in 1929. Through a special arrangement, he was able to join Bell Labs in April 1929, prior to finishing school. He would have been one of Harold Black's trusted colleagues, working in the same departments. It is certain that Black taught Swartzel Jr. the concepts of Negative Feedback firsthand.

K. D. Swartzel Jr. in 1934

Retro Electro Fun Fact: Following WWII, K. D. Swartzel Sr. was a senior scientist in the physics department at Cornell University, where he was involved in the post-war atomic bomb projects.

K. D. Swartzel Jr: designer of the summing amplifier – the first operational amplifier

At Bell Labs, Swartzel was surrounded by amplifiers that were now powerful but unruly and inconsistent. They could deliver enormous gain, but only if tightly controlled. Swartzel realized that this control could be turned into something more than clean amplification.

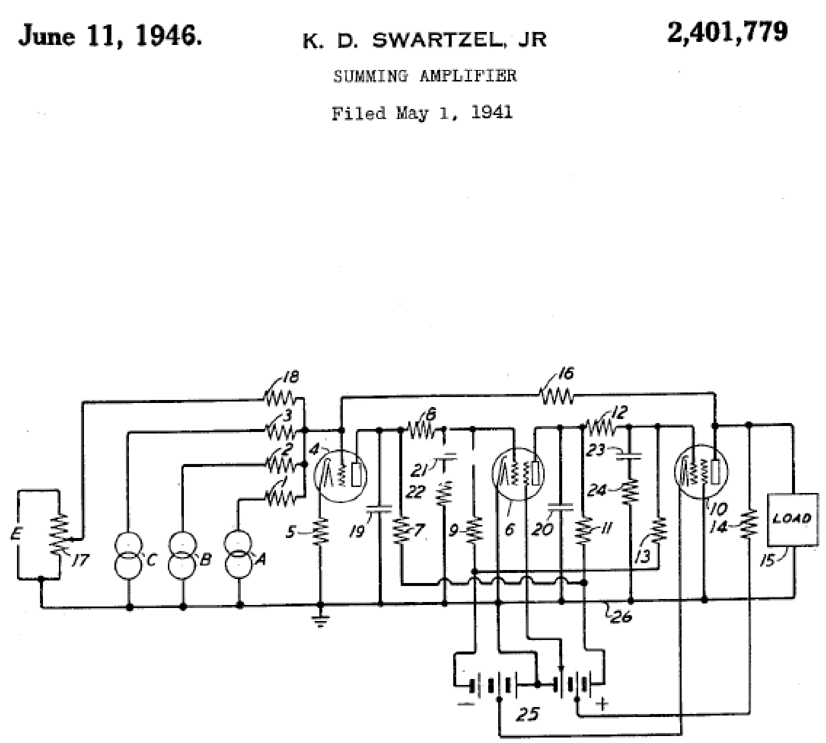
In 1941, he filed U.S. Patent 2,401,779 for what he called a summing amplifier.

At first glance, Swartzel's 1941 summing amplifier looks like a conventional multi-stage vacuum-tube amplifier. There are several triode stages, coupling networks, bias resistors, and a final output feeding a load. Nothing about the tubes themselves is exotic.

The novelty lies at the far left of the diagram.

Multiple independent input sources, labeled A, B, and C, each feed the amplifier through their own resistors. Those resistors converge at a single node on the control grid of the first tube. This junction is the heart of the invention. Instead of one input dominating the amplifier, each source supplies a current proportional to its voltage and resistor value.

Negative feedback from the output is routed back through a carefully chosen resistance path. The amplifier adjusts its output voltage until the input currents satisfy the constraints imposed by the feedback resistor network. The tubes themselves are not responsible for accuracy. They merely provide enough gain for feedback to be in control.

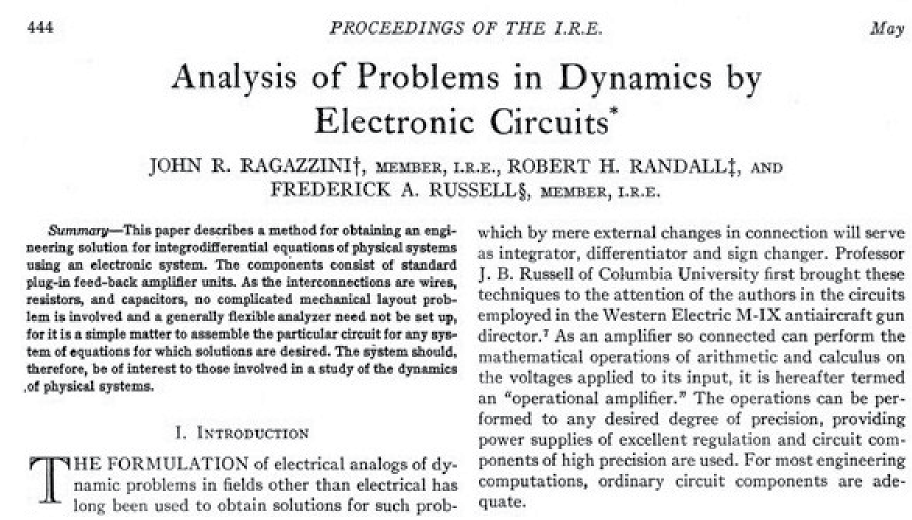


US Patent 2401779 "The Summing Amplifier"

As a result, the output voltage becomes a linear combination of the inputs. Each input's contribution is determined entirely by resistor ratios, not by tube characteristics, operating point, or drift. Precision is assigned entirely to passive high-precision components. The active devices are reduced to obedient workers. The internal operation of the amplifier is deliberately controlled by external constraints.

The summing amplifier arrived at exactly the right moment in 1941 at the start of WWII. Radar, fire-control systems, and early analog computing all needed reliable ways to combine signals continuously and in real time. Swartzel's circuit provided a building block that could be reused, scaled, and trusted.

Retro Electro fun fact: At the same time as Swartzel is filing for this patent in early 1941, Bell Labs would hire star rookie Claude Shannon out of MIT. Shannon would work with Swartzel's summing amplifiers to develop the earliest digital computers. Shannon was also part of the Dartmouth Summer Research Project on Artificial Intelligence. You can read that story in the Retro Electro Article "Programming a Calculator to Form Concepts." (www.digikey.com/en/emedia/emagazine/2024/edge-ai?page=11)



444 *PROCEEDINGS OF THE I.R.E.* *May*

Analysis of Problems in Dynamics by Electronic Circuits*

JOHN R. RAGAZZINI†, MEMBER, I.R.E., ROBERT H. RANDALL‡, AND FREDERICK A. RUSSELL§, MEMBER, I.R.E.

Summary—This paper describes a method for obtaining an engineering solution for integrodifferential equations of physical systems using an electronic system. The components consist of standard plug-in feed-back amplifier units. As the interconnections are wires, resistors, and capacitors, no complicated mechanical layout problem is involved and a generally flexible analyzer need not be set up, for it is a simple matter to assemble the particular circuit for any system of equations for which solutions are desired. The system should, therefore, be of interest to those involved in a study of the dynamics of physical systems.

I. INTRODUCTION

THE FORMULATION of electrical analogs of dynamic problems in fields other than electrical has long been used to obtain solutions for such prob-

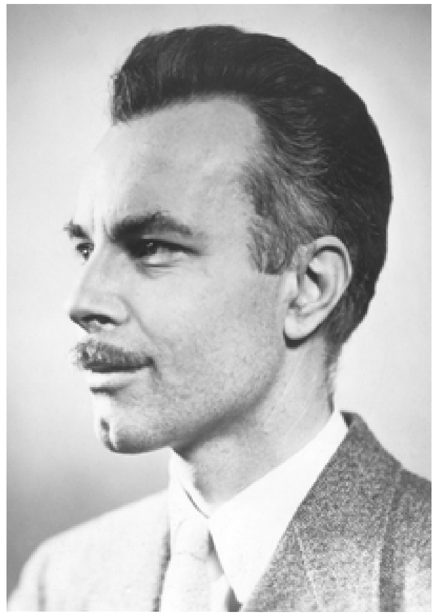
which by mere external changes in connection will serve as integrator, differentiator and sign changer. Professor J. B. Russell of Columbia University first brought these techniques to the attention of the authors in the circuits employed in the Western Electric M-IX antiaircraft gun director.[7] As an amplifier so connected can perform the mathematical operations of arithmetic and calculus on the voltages applied to its input, it is hereafter termed an "operational amplifier." The operations can be performed to any desired degree of precision, providing power supplies of excellent regulation and circuit components of high precision are used. For most engineering computations, ordinary circuit components are adequate.

Journal Article by Ragazzini where he coins the term "Operational Amplifier."

A few years later, John R. Ragazzini, an EE professor at Columbia University, would coin the name 'operational amplifier', so called because it could perform mathematical operations, like addition, subtraction, integration, and differentiation, by design.

George A. Philbrick: designer of the K2 – first practical retail operational amplifier

George Arthur Philbrick was born in 1913 in Belmont, Massachusetts, at a time when electrical engineering was quickly being recognized as a profession rather than a simple trade or hobby. By his college years, vacuum tubes had moved beyond radios and started appearing in laboratories, factories, and telephone exchanges. Philbrick was more interested in systems than gadgets, believing that circuits could be built, controlled, and analyzed much like mathematical entities.



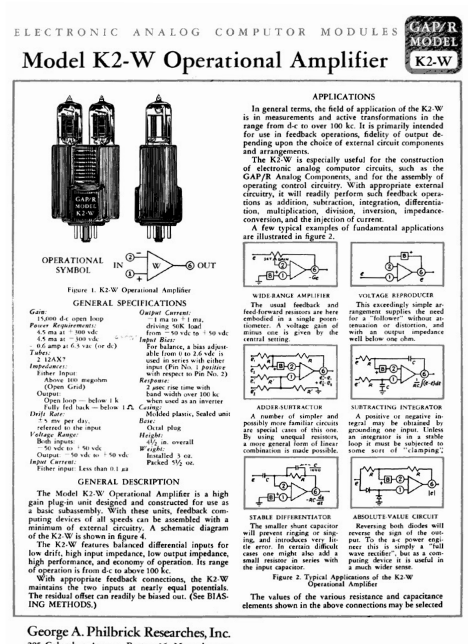
George Arthur Philbrick

He enrolled at Columbia University, where he studied electrical engineering during the 1930s, a period when Columbia stood at the center of American control theory and communications research. The university maintained close ties to Bell Labs, Western Electric, and the wartime research ecosystem forming around New York City. More importantly, Columbia faculty, such as John R. Ragazzini, were redefining what amplifiers could be used for. They were no longer just signal boosters, they were now be used as computational elements. While earlier engineers focused on tube linearity and component perfection, Columbia University's approach highlighted a system of rules: feedback, summation, integration, and constraint.

During World War II, Philbrick directly applied these concepts. Like many engineers of his generation, he was pulled into classified work involving radar, control systems, and high-speed analog computation. These systems demanded repeatable behavior under stress, temperature variations, and the presence of imperfect components. By the end of the war, Philbrick had firsthand experience with operational amplifiers not as abstractions but as working tools embedded in real machines.

At that time, operational amplifiers existed mostly as schematics on paper, custom-built rack setups, or one-off laboratory oddities. Each system needed skilled engineers to design, wire, troubleshoot, and maintain them. There was no standard module, no catalog part, and no "box" you could simply order that would reliably perform an operation. Precision electronics remained confined behind laboratory doors.

By the end of World War II, the operational amplifier was well understood. Engineers at Bell Labs and Columbia University demonstrated that feedback-controlled amplifiers could perform addition, subtraction, integration, differentiation, and reliably stabilize complex systems. What once were delicate laboratory experiments had become proven wartime tools. The question had shifted *from* whether these amplifiers worked to how engineers could get their hands on them. That was the gap Philbrick recognized when he started his new business.



ELECTRONIC ANALOG COMPUTOR MODULES

GAP/R MODEL K2-W

Model K2-W Operational Amplifier

OPERATIONAL SYMBOL

Figure 1. K2-W Operational Amplifier

GENERAL SPECIFICATIONS

Gain: 15,000 d-c open loop

Power Requirements: 4.5 ma at +300 vdc; 4.5 ma at −300 vdc; 0.6 amp at 6.3 vac (or dc)

Tubes: 2 12AX7

Impedances: Either Input: Above 100 megohm (Open Grid). Output: Open loop — below 1 k; Fully fed back — below 1 Ω

Drift Rate: ±5 mv per day, referred to the input

Voltage Range: Both inputs: −50 vdc to +50 vdc. Output: −50 vdc to +50 vdc

Input Current: Either input: Less than 0.1 μa

Output Current: −1 ma to +1 ma, driving 50K load from −50 vdc to +50 vdc

Input Bias: For balance, a bias adjustable from 0 to 2.6 vdc is used in series with either input (Pin No. 1 *positive* with respect to Pin No. 2)

Response: 2 μsec rise time with band width over 100 kc when used as an inverter

Casing: Molded plastic, Sealed unit

Base: Octal plug

Height: 4½ in. overall

Weight: Installed 3 oz. Packed 5½ oz.

GENERAL DESCRIPTION

The Model K2-W Operational Amplifier is a high gain plug-in unit designed and constructed for use as a basic subassembly. With these units, feedback computing devices of all speeds can be assembled with a minimum of external circuitry. A schematic diagram of the K2-W is shown in figure 4.

The K2-W features balanced differential inputs for low drift, high input impedance, low output impedance, high performance, and economy of operation. Its range of operation is from d-c to above 100 kc.

With appropriate feedback connections, the K2-W maintains the two inputs at nearly equal potentials. The residual offset can readily be biased out. (See BIASING METHODS.)

APPLICATIONS

In general terms, the field of application of the K2-W is in measurements and active transformations in the range from d-c to over 100 kc. It is primarily intended for use in feedback operations, fidelity of output depending upon the choice of external circuit components and arrangements.

The K2-W is especially useful for the construction of electronic analog computor circuits, such as the GAP/R Analog Components, and for the assembly of operating control circuitry. With appropriate external circuitry, it will readily perform such feedback operations as addition, subtraction, integration, differentiation, multiplication, division, inversion, impedance-conversion, and the injection of current.

A few typical examples of fundamental applications are illustrated in figure 2.

WIDE-RANGE AMPLIFIER

The usual feedback and feed-forward resistors are here embodied in a single potentiometer. A voltage gain of minus one is given by the central setting.

VOLTAGE REPRODUCER

This exceedingly simple arrangement supplies the need for a "follower" without attenuation or distortion, and with an output impedance well below one ohm.

ADDER-SUBTRACTOR

A number of simpler and possibly more familiar circuits are special cases of this one. By using unequal resistors, a more general form of linear combination is made possible.

SUBTRACTING INTEGRATOR

A positive or negative integral may be obtained by grounding one input. Unless an integrator is in a stable loop it must be subjected to some sort of "clamping".

STABLE DIFFERENTIATOR

The smaller shunt capacitor will prevent ringing or singing, and introduces very little error. In certain difficult cases one might also add a small resistor in series with the input capacitor.

ABSOLUTE-VALUE CIRCUIT

Reversing both diodes will reverse the sign of the output. To the a-c power engineer this is simply a "full wave rectifier", but as a computing device it is useful in a much wider sense.

Figure 2. Typical Applications of the K2-W Operational Amplifier

The values of the various resistance and capacitance elements shown in the above connections may be selected

George A. Philbrick Researches, Inc.
285 Columbus Avenue, Boston 16, Massachusetts

GAP/R K2-W Op Amp Datasheet

Philbrick figured that the next technological revolution wouldn't stem from inventing new mathematics or tubes, but from packaging trustworthy modular components to build that new technology. If amplifiers could be made modular, calibrated, documented, and repeatable, then the techniques developed at Columbia and Bell Labs could spread into factories, universities, and engineering offices everywhere.

In 1946, driven by this belief, George Philbrick established George A. Philbrick Researches, Inc. (Often abbreviated as GAP/R) in Boston. The company's goal was straightforward: to market operational amplifiers as standardized components for others to use. Essentially a development board in today's terms. By doing so, Philbrick shifted the operational amplifier from an academic idea to a commercial product with the launch of the K2 series of operational amplifiers in 1952, quietly paving the way for advancements in analog computing, control engineering, and the modular electronics industry.

The amplifier had finally become trustworthy enough to put in a box, and Philbrick was the one who put it there.

Over the years, GAP/R has been bought and traded several

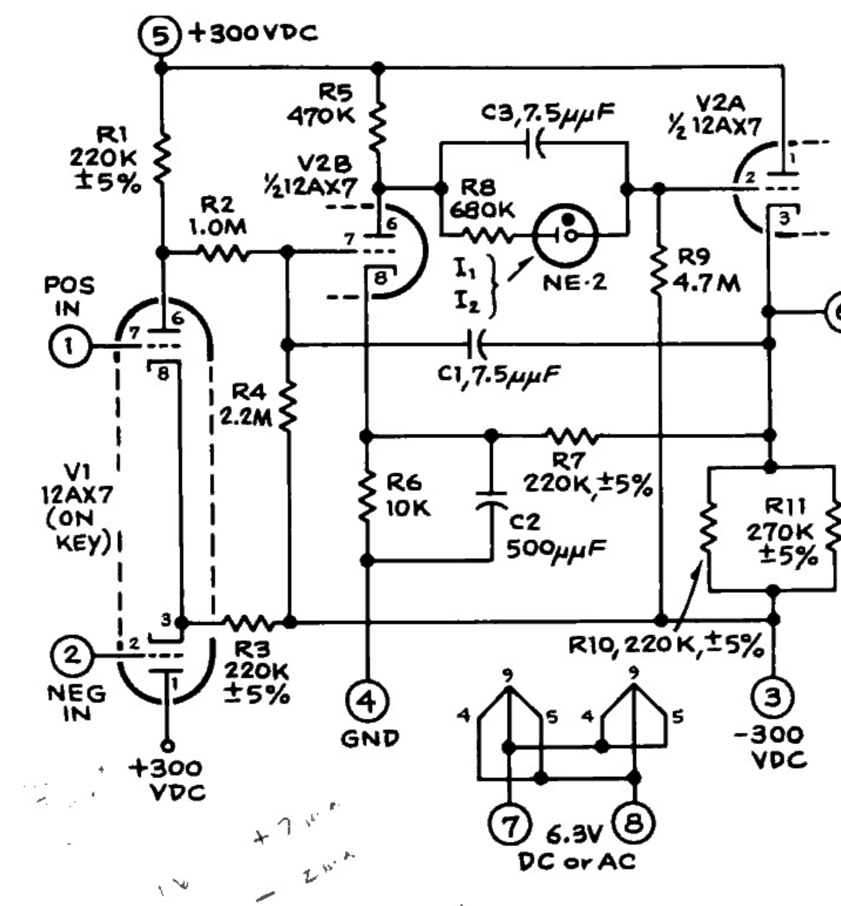
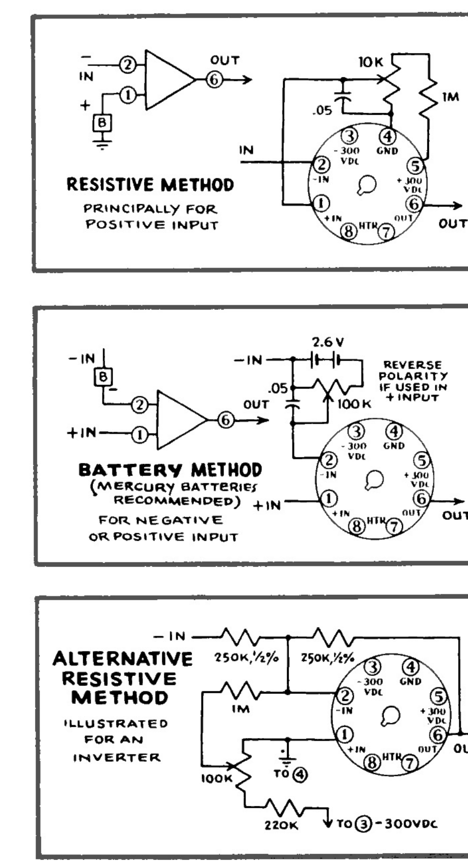
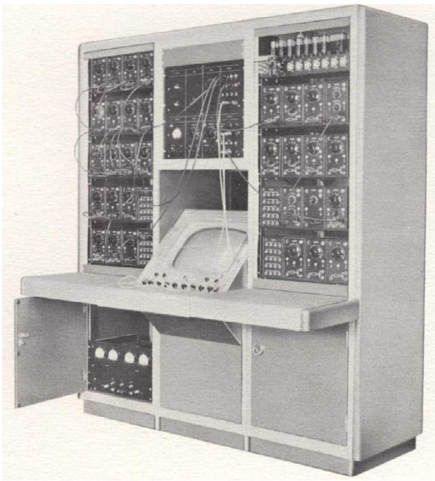


Figure 4. Schematic Diagram



Typical Application Notes for the K2-W.



Early 'analog computer' using Philbrick's vacuum tube-based op-amps.

times. It appears that its lineage now leads to it being owned by **Microchip**.

Transistors/post-vacuum tubes

In the 1950s and early 1960s, as transistors were first being used widely, engineers would fashion equivalent operational amplifier circuits from discrete transistors, not unlike Swartzel's summing amplifier with discrete vacuum tubes.

These transistorized designs reduced size, power consumption, and warm-up time, but they did not fundamentally change how op amps were conceived or used. They were still assembled by hand, still sensitive to device variation, and still the domain of specialists.

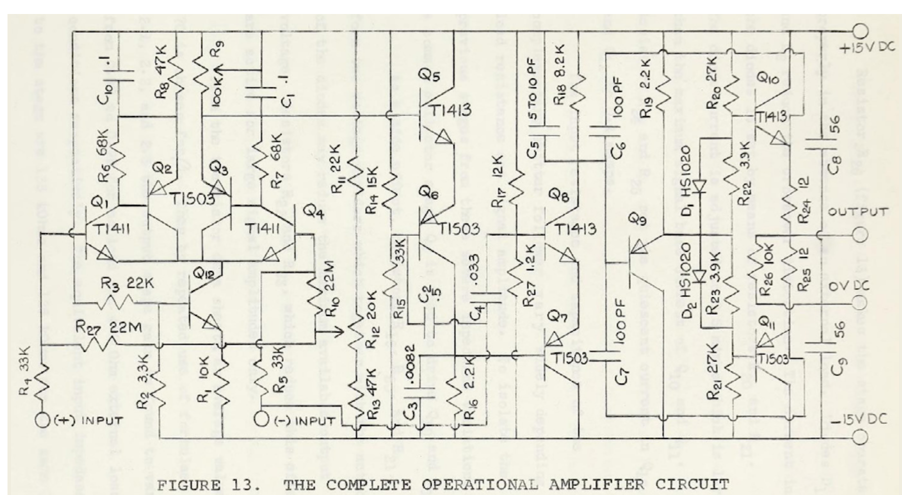
The op amp had survived the transition from tubes to transistors, but it still wasn't manufacturable at scale.

Bob Widlar: designer of the μA703, μA709, and LM101

Robert John Widlar was born on November 30, 1937, in Cleveland, Ohio. He graduated from the University of Colorado at Boulder and, in the late 1950s, joined the United States Air Force as an instructor, teaching servicemen about electronic equipment. One of the classes he wrote and taught, 'Introduction to Semiconductor Devices,' is available online.

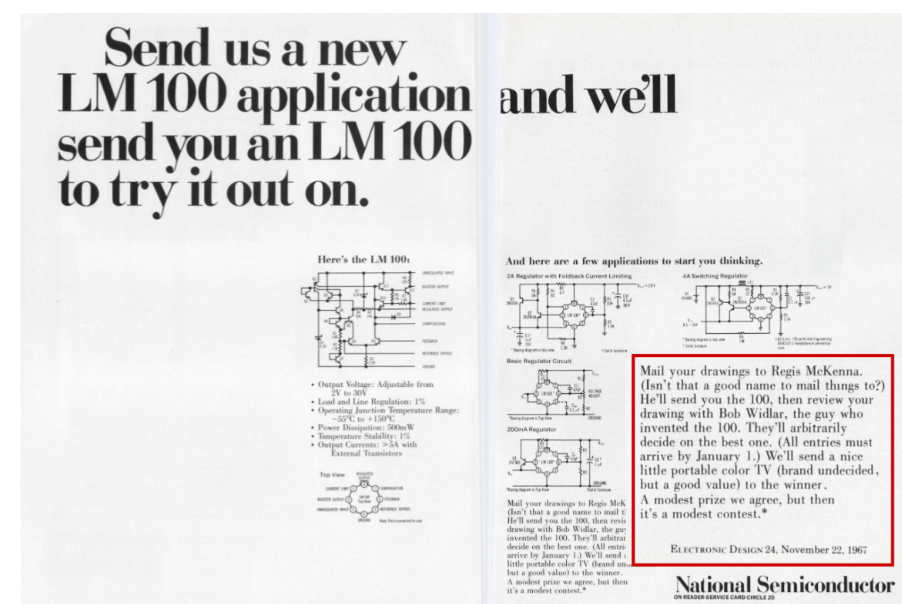
Following a brief Air Force enlistment, Bob Widlar joined **Fairchild Semiconductor** in 1961. That same year, three Fairchild engineers, **Lionel Kattner**, **David Allison**, and **David James**, left to start a company focused on integrated circuit design named **Signetics**. At the time, Fairchild was primarily focused on improving transistors and didn't prioritize integrated circuits, but the formation of Signetics marked the beginning of a shift in the company's focus toward more integrated circuit design.

Widlar's first year at Fairchild was fairly mundane, from how it sounds. His tasks focused around biasing, current sources, and manufacturable linear circuits, learning how to make unreliable silicon transistors



Schematic of transistor based operational amplifier.

Bob Widlar at National Semiconductor in 1967.



Have your drawings reviewed personally by Bob Widlar.

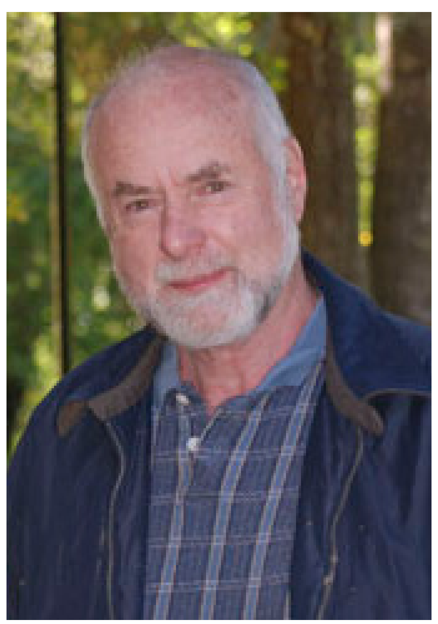
behave predictably. He learns how badly real transistors drift, mismatch, and lie. He would write documentation and references for other engineers. He soon got into integrated circuit design.

In 1963, Widlar designed the first monolithic operational amplifier, the μA702. At a price of $300 a piece, the primary customer would have been the military and government. The next year, he designed the improved μA709, which retailed for $70. It was a wild success. Fairchild now had an op-amp the size of a contemporary transistor before anybody else. Lore has it that when the μA709 was released, it was the first Integrated circuit to have universal utility across every industry, and as prices dropped, it quickly became the highest-selling IC in history, at that time.

Following the success of this new op-amp at Fairchild, Widlar wanted a raise. When denied this raise, he quit and went to work for **National Semiconductor**. In 1967, he created an even better op-amp with the LM101. National Semiconductor was so confident in the LM101 that they ran large magazine ads for a contest where the prize was to have Widlar perform a design review of your drawings, and the grand prize was a color TV.

David Fullagar: designer of the LM741 – the gold standard for Op-Amps

As Widlar was leaving Fairchild, a young British immigrant named **David Fullagar** joined Fairchild. In an interview with the Computer History Museum, Fullagar describes his first days at Fairchild Semiconductor as *"My initial assignment was: 'Widlar left on Friday (to join National Semiconductor), I showed up on Monday. The yield for the μA709 was one die per wafer. Could I fix it?' So, I spent some number of weeks sort of analyzing the 709, trying to figure out what was going on, and came to the conclusion that there was nothing wrong with the circuit. And it turned out the yield problem was that they'd switched DI (deionized) water suppliers at the factory, and the new DI water they were using wasn't of the appropriate purity or something like that, so it was an entirely independent problem."*

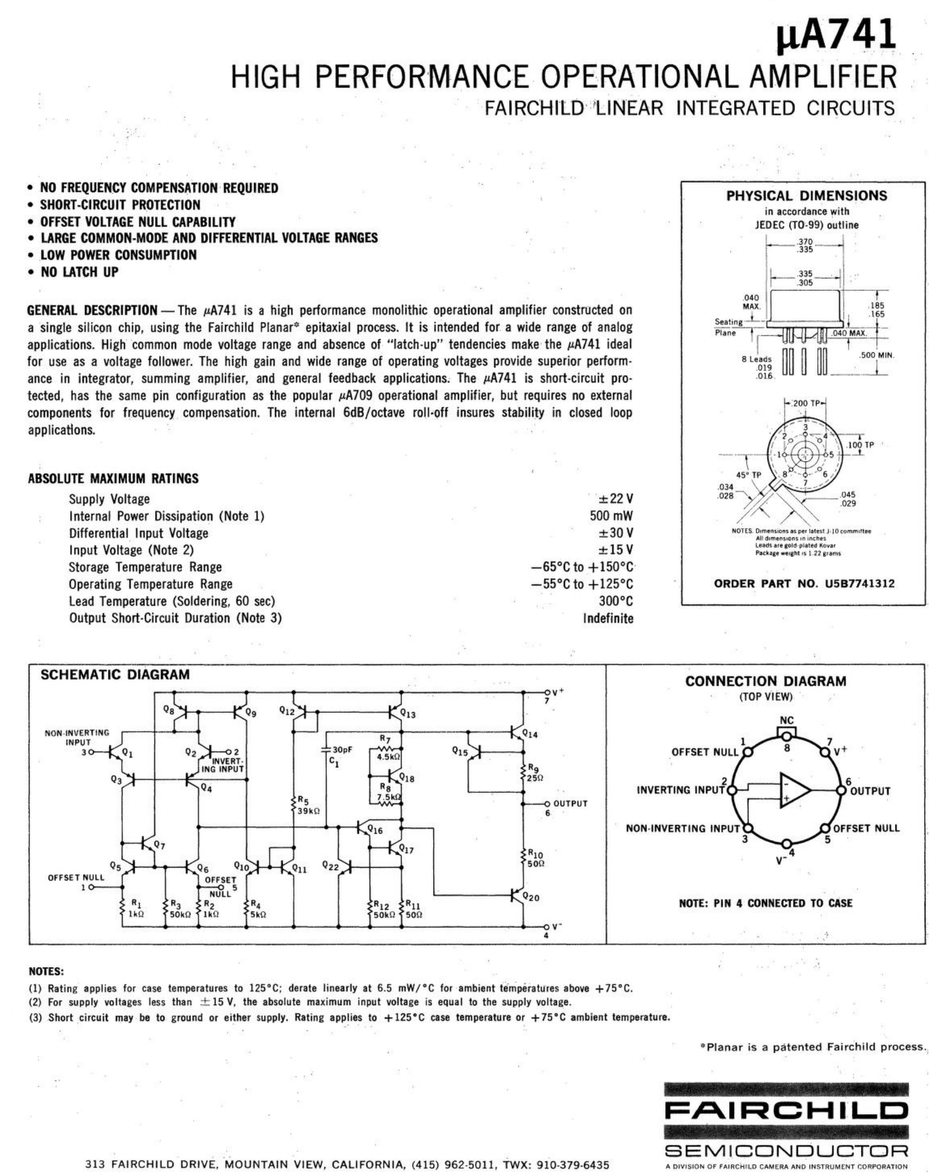


David Fullagar

Retro Electro fun fact: It is said that the μA741 op-amp is the second-best-selling integrated circuit in history, behind the 555 timer. To read the full untold story of the 555 timer, check out the Retro Electro article "Five Five Five: The Story of Interdesign Inc." [(www.digikey.com/en/emedia/emagazine/2024/plcs?page=16)](www.digikey.com/en/emedia/emagazine/2024/plcs?page=16)

In 1967, when National Semiconductor released the LM101, Fullagar bought some of them to check out. He found a flaw in the input stage and used that new knowledge to create an improved version of National's LM101 for Fairchild, known as the μA741.

In the late 1960s, a mass exodus of talent left Fairchild. As **Gordon Moore** and **Bob Noyce** left to found Intel, they offered Fullagar a position, but he and others instead joined **Intersil**, which was also founded by Fairchild alumni. In the early 1980s, Intersil was acquired by **General Electric**, prompting Fullagar and a couple of his coworkers, **Jack Gifford** and **Fred Beck**, to start **Maxim Integrated**. While serving as VP of Engineering at Maxim, the company grew from a startup of three engineers to one generating half a billion dollars in annual revenue.



μA741

HIGH PERFORMANCE OPERATIONAL AMPLIFIER

FAIRCHILD LINEAR INTEGRATED CIRCUITS

- NO FREQUENCY COMPENSATION REQUIRED
- SHORT-CIRCUIT PROTECTION
- OFFSET VOLTAGE NULL CAPABILITY
- LARGE COMMON-MODE AND DIFFERENTIAL VOLTAGE RANGES
- LOW POWER CONSUMPTION
- NO LATCH UP

GENERAL DESCRIPTION — The μA741 is a high performance monolithic operational amplifier constructed on a single silicon chip, using the Fairchild Planar* epitaxial process. It is intended for a wide range of analog applications. High common mode voltage range and absence of "latch-up" tendencies make the μA741 ideal for use as a voltage follower. The high gain and wide range of operating voltages provide superior performance in integrator, summing amplifier, and general feedback applications. The μA741 is short-circuit protected, has the same pin configuration as the popular μA709 operational amplifier, but requires no external components for frequency compensation. The internal 6dB/octave roll-off insures stability in closed loop applications.

ABSOLUTE MAXIMUM RATINGS

Supply Voltage	±22 V
Internal Power Dissipation (Note 1)	500 mW
Differential Input Voltage	±30 V
Input Voltage (Note 2)	±15 V
Storage Temperature Range	−65°C to +150°C
Operating Temperature Range	−55°C to +125°C
Lead Temperature (Soldering, 60 sec)	300°C
Output Short-Circuit Duration (Note 3)	Indefinite

PHYSICAL DIMENSIONS in accordance with JEDEC (TO-99) outline

ORDER PART NO. U5B7741312

SCHEMATIC DIAGRAM

CONNECTION DIAGRAM (TOP VIEW)

NOTE: PIN 4 CONNECTED TO CASE

NOTES:
(1) Rating applies for case temperatures to 125°C; derate linearly at 6.5 mW/°C for ambient temperatures above +75°C.
(2) For supply voltages less than ±15 V, the absolute maximum input voltage is equal to the supply voltage.
(3) Short circuit may be to ground or either supply. Rating applies to +125°C case temperature or +75°C ambient temperature.

*Planar is a patented Fairchild process.

313 FAIRCHILD DRIVE, MOUNTAIN VIEW, CALIFORNIA, (415) 962-5011, TWX: 910-379-6435

FAIRCHILD SEMICONDUCTOR
A DIVISION OF FAIRCHILD CAMERA AND INSTRUMENT CORPORATION

Bob Widlar at National Semiconductor in 1967.

Why the μA741 still ships today

By modern standards, the μA741 should have disappeared long ago.

It is slow, noisy, power-hungry, and incapable of rail-to-rail operation. No engineer designing a high-performance circuit would reach for it first. Yet more than fifty years after its introduction, the μA 741 remains in production, not because it excels, but because it behaves.

The μA 741 solved a problem more important than performance: predictability. Earlier integrated op amps worked only if designers understood their quirks and compensated for them carefully. The μA741 changed that expectation. It was stable, tolerant of abuse, and difficult to misuse. Its behavior was consistent from lab bench to factory floor, making it ideal for education, documentation, and standardized designs. Once textbooks, lab manuals, and training programs adopted it, the μA741 became less a component than a shared reference point.

Decades of field history, known failure modes, and well-understood behavior reduce risk in ways no datasheet can capture. The μA741 persists not out of nostalgia, but because it embodies the original promise of the operational amplifier: an obedient, predictable building block that quietly enforces the rules and stays out of the way.

Black made amplifiers trustworthy.

Swartzel showed that trusted amplifiers can do math.

Ragazzini created the rules for the math.

Philbrick turned the operational amplifier into a module and a business.

Widlar makes it mass manufacturable in silicon.

Fullagar made it clean and created the gold standard for operational amplifiers.

In 2015, Fairchild Semiconductor was acquired by On Semiconductor for $2.4B. As a result, the closest modern descendant of the original Fairchild µA741 is ON Semiconductor's LM741, but at some point in the past several years, OnSemi discontinued its LM741 line of op-amps, but there is no shortage of other manufacturers, like Texas Instruments, that still produce the LM741 to this day. (Link: https://www.digikey.com/en/products/detail/texas-instruments/LM741CN-NOPB/6322)

Suggested Reading

"Chip Hall of Fame: Fairchild Semiconductor µA741 Op-Amp" by IEEE Spectrum

The George A. Philbrick Researches Archive

"Unsung Hero Pioneered Op Amp" by Design Classics

"Op Amp History" by OldHackee3915 on YouTube

Applications Manual for Philbrick Octal Plug-In Computing Amplifiers

"The Operational Amplifier" by Columbia University

"600 Volt Tube Op Amp! Lets Power It Up!" by Mr Carlson's Lab

"Analog Computers and Synths" by Sequence 15

"The Oral History of David Fullagar" by the Computer History Museum

"Bob Widlar – The Life of an Engineering Legend" by Sam Sattel

"The 741 Op Amp" by Tiny Transistors

"Dave Fullagar, Analog-IC Designer and Entrepreneur" by Computer History Museum

1927

Harold S. Black conceives the negative feedback amplifier, realizing that distortion can be canceled by feeding output back in opposition to the input.

1929

Karl D. Swartzel Jr. joins Bell Laboratories while completing his mathematics education, entering an engineering culture shaped by Black's feedback work.

1937

Black's negative feedback patent is finally granted after years of skepticism, formalizing feedback as a controllable engineering principle.

1941

Swartzel files U.S. Patent 2,401,779 for the summing amplifier, demonstrating that feedback-controlled amplifiers can perform mathematical operations.

Claude Shannon joins Bell Labs; summing amplifiers become part of the broader environment supporting early digital and analog computation.

Mid-1940s

John R. Ragazzini coins the term operational amplifier, defining the device by the mathematical operations it performs rather than its construction.

1946

George A. Philbrick founds George A. Philbrick Researches, Inc. (GAP/R), recognizing the need for standardized, modular operational amplifiers.

1952

Philbrick introduces the K2-W, the first commercially available general-purpose operational amplifier module.

1950s

Discrete-transistor operational amplifiers slowly replace vacuum tubes, reducing size and power while preserving the same feedback-based concepts.

1963

Harold Bob Widlar designs the µA702 at Fairchild Semiconductor, the first monolithic integrated-circuit operational amplifier.

1964

Widlar follows with the µA709, dramatically lowering cost and proving integrated op amps can serve general-purpose roles.

1967

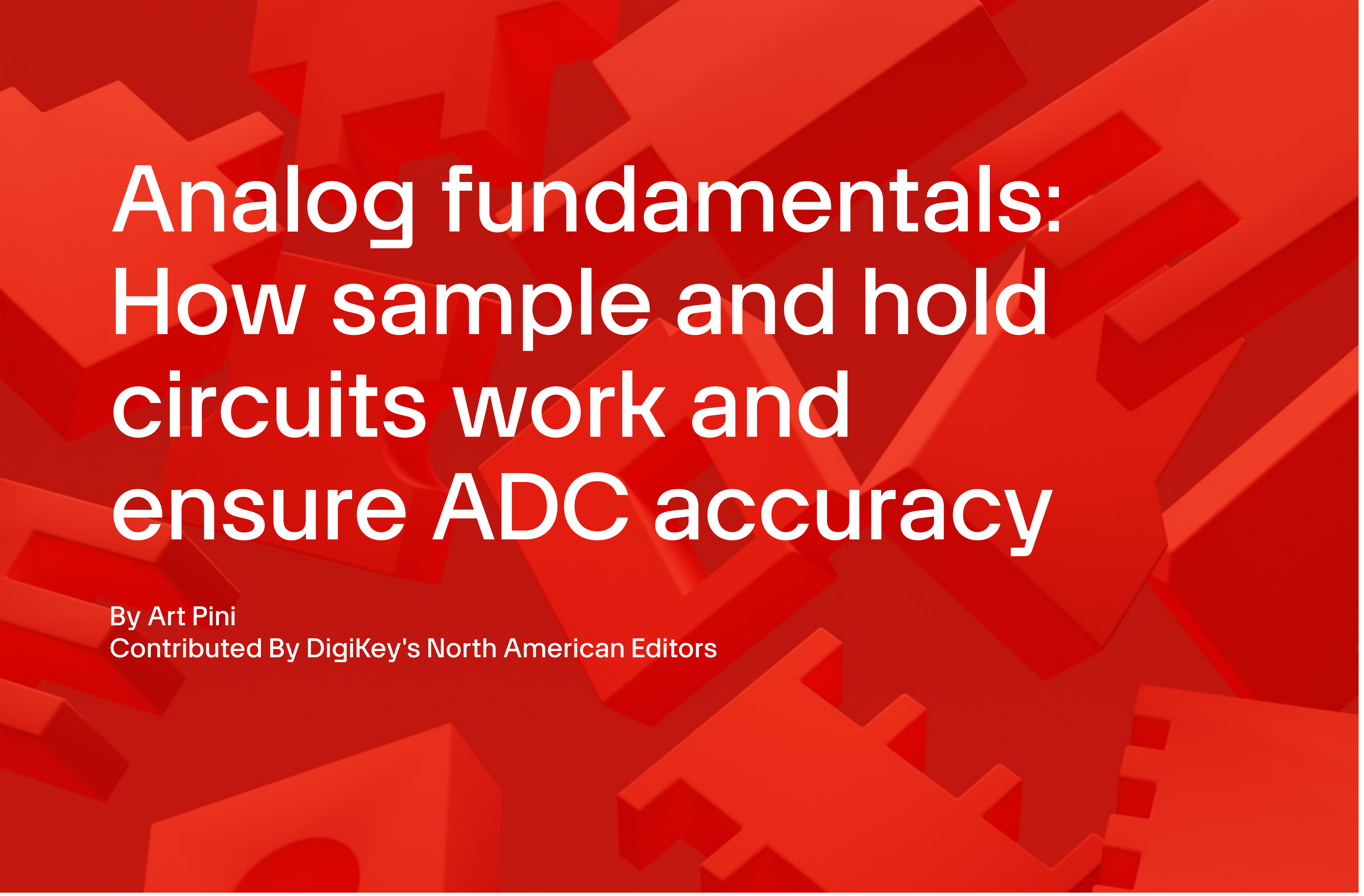
Widlar designs the LM101 at National Semiconductor, improving stability and usability through compensation techniques.

1968

David Fullagar designs the µA741 at Fairchild Semiconductor, creating a forgiving, internally compensated op amp that becomes the industry standard.

Present Day

The µA741 remains in production, valued for predictability, educational clarity, and decades of proven behavior.



Analog fundamentals: How sample and hold circuits work and ensure ADC accuracy

By Art Pini
Contributed By DigiKey's North American Editors

Converting an analog signal from the "real" world to a digital signal that can be processed upstream is a fundamental function of electronic systems, ranging from audio recording to the Internet of Things (IoT), the Industrial IoT (IIoT), and now the artificial intelligence (AI) of Things (AIoT). However, to perform it effectively and efficiently requires a level of understanding of the underlying principles and steps that are often overlooked.

For instance, how exactly is a signal "held" and then "sampled" before conversion, given that a typical analog signal applied to the input of an analog-to-digital converter (ADC) is changing amplitude continuously and will be different at the end of the conversion than it was at the beginning? This amplitude change or skewing can result in a serious error, especially for high resolution ADCs which take more time to convert a signal. The challenge for designers is to both understand and eliminate this source of error.

This article shows how preventing amplitude skew is accomplished using a sample and hold (S&H) or track and hold (T&H) circuit for the ADC. The S&H (or T&H) performs the true sampling of the input and operates between the input anti-aliasing low-pass filter and the ADC. The article discusses the characteristics and selection criteria for S&H ICs and looks at ADCs with integrated S&Hs. Example devices with varying characteristics for different applications from [Texas Instruments](), Maxim Integrated, and Analog Devices are used for illustrative purposes.

The role of sample-and-hold in ADCs

When a non-DC signal is applied to the input of an ADC, it is changing amplitude continuously. However, the analog-to-digital conversion process takes a finite interval of time, so over that time, the amplitude of the ADC input will change (Figure 1). It is this amplitude skewing that results in a potentially serious error.

Preventing amplitude skew in an ADC is a matter of sampling the signal and holding a fixed amplitude while the conversion is in process. This is accomplished using the S&H or T&H circuit for the ADC (Figure 2).

Both types of circuits sample the input signal and hold the sampled voltage constant for the duration of the conversion process. The T&H circuit output (right) tracks the input signal until signaled to sample; it then holds the sample value during the ADC conversion. The S&H has a shorter sample aperture and its output is a series of sampled levels (left). The key difference between T&H and S&H is the duration of the tracking interval: very short for the S&H and significantly longer for the T&H. Both circuits rely on a fast switch to isolate a storage capacitor that has been connected to the signal input. The balance of this article will use S&H synonymously with S&H or T&H.

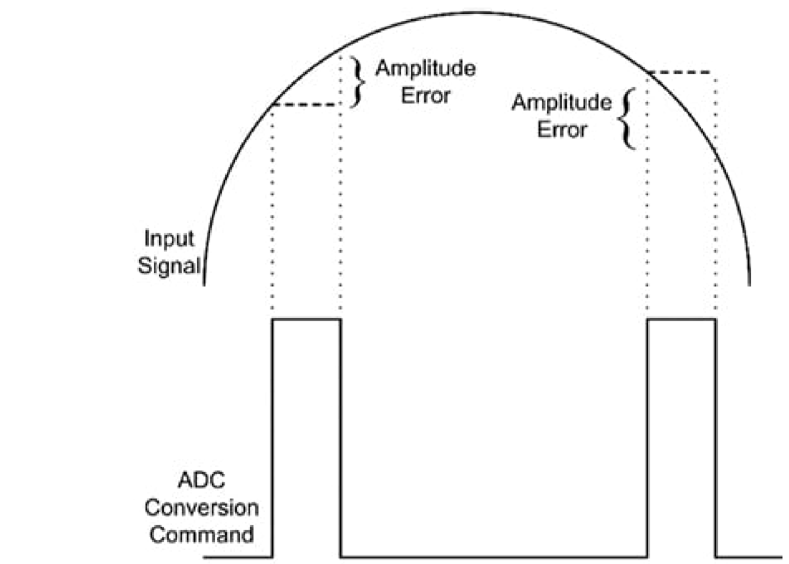


Figure 1: An ADC with a varying input signal is subject to amplitude errors (top) due to the signal amplitude variations during digitization (bottom). (Image source: DigiKey)

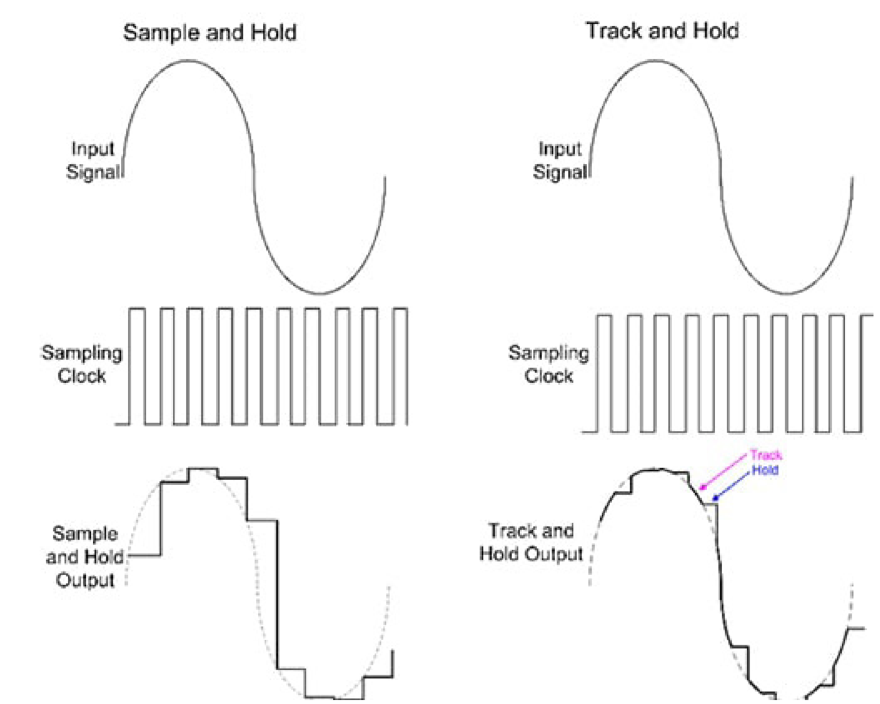


Figure 2: The primary difference between an S&H (left) and a T&H (right) circuit is the duration of the tracking period: it is short in the S&H and long in the T&H. (Image source: DigiKey)

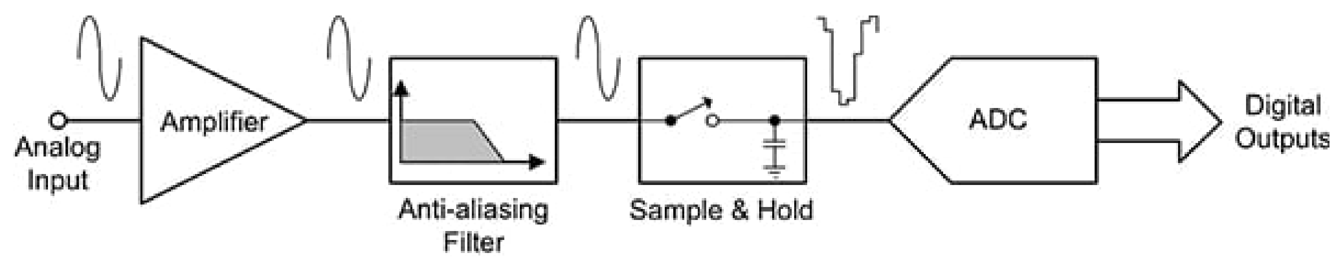


Figure 3: In the digitizer signal path, the S&H is placed between the anti-aliasing low-pass filter and the ADC. (Image source: DigiKey)

The S&H stage performs the true sampling of the input and operates between the input anti-aliasing low-pass filter and the ADC. The low-pass filter performs anti-aliasing band limiting and must precede the S&H to bandlimit the signal before sampling to prevent aliasing (Figure 3).

Note that the signals before the S&H are all analog signals. The output of the S&H is a sampled waveform that is fed to the ADC.

A typical S&H device

The block diagram of the Texas Instruments [LF398MX/NOPB](https://www.digikey.com) S&H integrated circuit (IC) shows the basic circuit configuration (Figure 4). The S&H is implemented using a fast switch and a high quality capacitor. In the case of the LF398MX/NOPB, the capacitor is external to the IC. When the switch is closed, the capacitor is charged to the input signal voltage level. When the switch is open, the capacitor retains that voltage until it is digitized

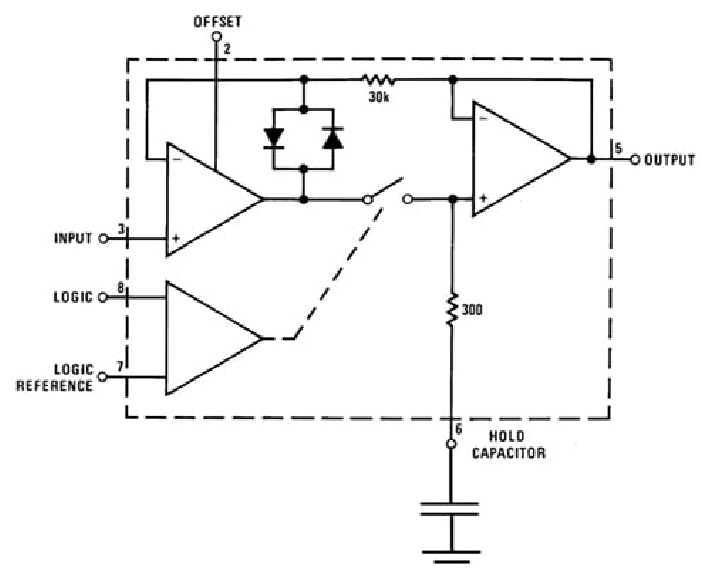


Figure 4: The block diagram of the Texas Instruments LF398MX/NOPB S&H shows the key components: a fast switch and an external hold capacitor. (Image source: Texas Instruments)

by the ADC. This S&H uses bi-FET technology, which combines FETs with bipolar transistors, to support fast acquisition (less than 6 microseconds (µs) with 0.01% amplitude error) with high DC accuracy (typically 0.002%), and an extremely low voltage droop (typically less than 83 microvolts (µV) per second). Internal amplifiers buffer the switch and hold capacitor.

The acquisition time of the S&H is dependent on the value of the hold capacitor, which can be in the range of 0.001 to 0.1 microfarad (µF). The external hold capacitor has to have low dielectric absorption and low leakage. Polystyrene, polypropylene, and teflon capacitors are recommended.

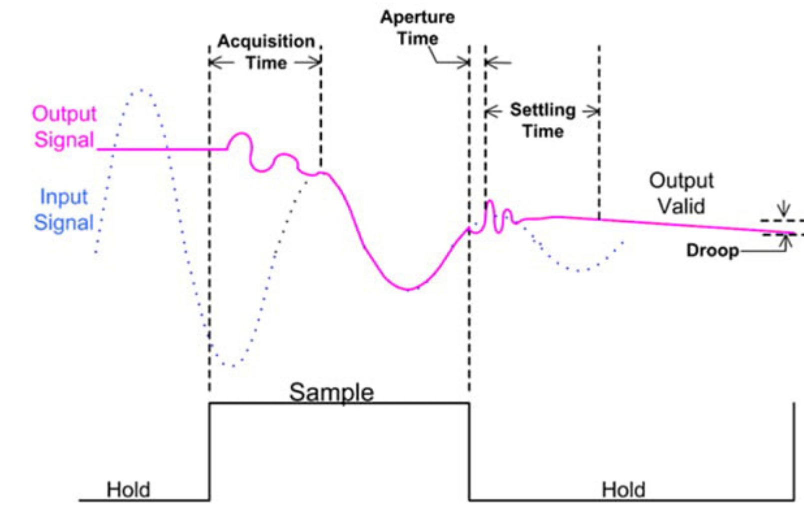


Figure 5: The definitions of common S&H dynamic characteristics include acquisition time, settling time, aperture time, and amplitude droop. (Image source: DigiKey)

S&H characteristics

S&H devices have a number of specific terms to describe their operation (Figure 5).

Acquisition time is the time from switching into sample mode until the S&H begins tracking the input signal. It is a function of the value of the hold capacitor and the series resistance of the switch and the signal path. When the mode returns to hold, there may be a time delay until the device stops tracking the input and begins to hold a value—this is the aperture time. Aperture time is a function of propagation delays of the drivers and the switch. Aperture uncertainty or jitter is the variation in the aperture time due to clock variations and noise.

Once in the hold mode, there is a time between entering that mode and when the device settles to within an error band about the hold value called the settling or hold settling time. Part of the settling time may include an unwanted transfer of charge between the switch driver and the hold capacitor; this is called the hold step or pedestal error. Hold step usually has magnitudes in the millivolt (mV) range and its effect is minimized by keeping the full-scale range of the signals as high as possible.

The shortest sampling period for the S&H is the sum of the acquisition time, aperture time, and the settling time.

The maximum sampling rate possible is the reciprocal of the sum of the acquisition time, aperture time, and settling time.

While in the hold mode, the S&H hold value may decrease due to leakage from the hold capacitor. This voltage increment is called droop. It is usually specified as a droop rate in mV per second.

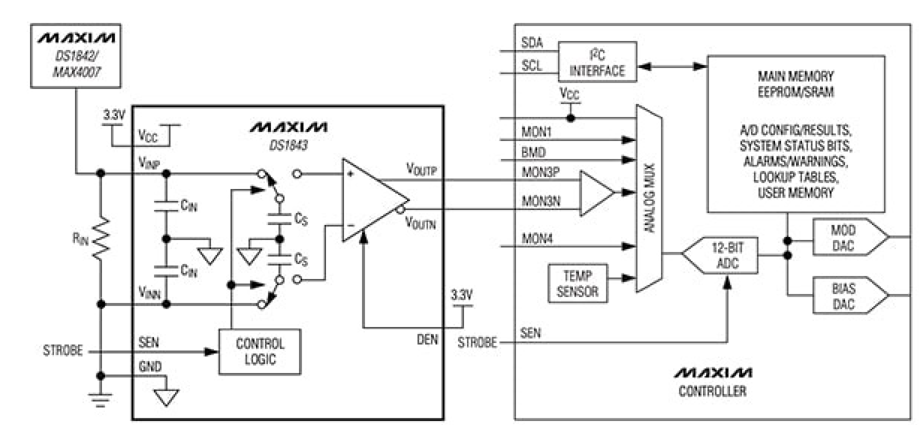


Figure 6: As shown in this typical operating circuit, the Maxim Integrated DS1843+TRL is a differential S&H that uses dual hold capacitors to implement differential sampling. (Image source: Maxim Integrated)

S&H configurations

S&H ICs are available in many configurations to match application needs. Consider an application requiring differential inputs such as interfacing with a differential output transducer like an accelerometer, strain gage, or optical current monitor. The Maxim Integrated DS1843D+TRL is a good example of an S&H IC for such applications (Figure 6).

The DS1843+TRL shown is in a typical optical line transmission application for burst mode received signal strength indicator (RSSI) measurement. The Maxim Integrated DS1842/MAX4007 is a current monitor that mirrors a current from an avalanche photodiode attached to its reference input. The output current is directed through the resistor, R_{IN}, converting it into a voltage. This voltage is measured differentially by the DS1843, which consists of fully differential sampling switches and capacitors, C_S, and a differential output buffer. This S&H uses two 5 picofarad (pF) capacitors, one connected to the positive differential input and the other connected to the negative differential input. The low capacitance value assures fast acquisition time. This device has a fast sample (acquisition) time of less than 300 nanoseconds (ns). The S&H's hold time is greater than 100 µs.

Devices are available that hold four or eight S&H circuits in a single IC package. An example is Analog Devices' SMP04ESZ-REEL quad S&H. The SMP04ESZ-REEL is a CMOS device incorporating four S&H circuits in a common package and features an acquisition time of 7 µs and a droop rate of only 2 mV/s (Figure 7).

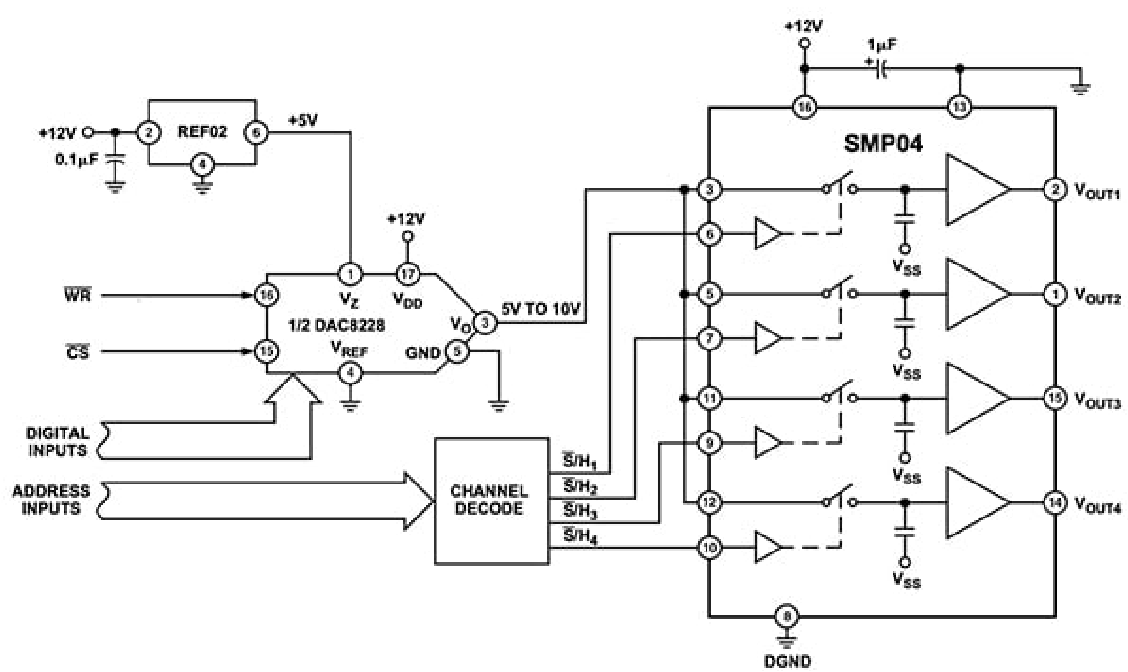


Figure 7: The Analog Devices SMP04 quad S&H contains four independent S&H circuits along with matching buffer amplifiers. The circuit shown uses the SMP04 being used to multiplex the output of a DAC into four channels. (Image source: Analog Devices)

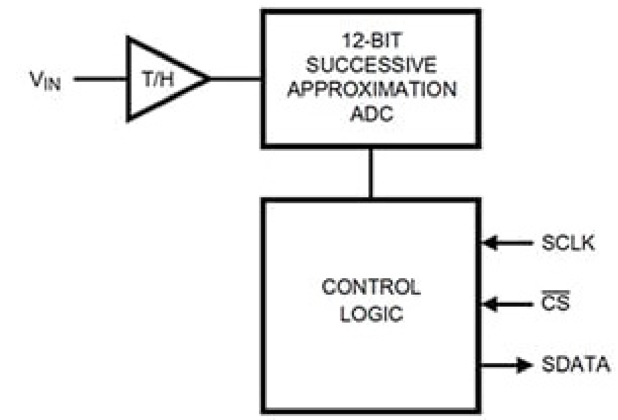


Figure 8: The Texas Instruments ADC121S021 is a 12-bit single-channel SAR ADC with a built-in T&H circuit. (Image source Texas Instruments)

Figure 7 also illustrates how S&Hs can be used with digital-to-analog converters (DACs), in this case to prevent output transients or glitches caused by code transitions in the DAC.

In the Figure, the SMP04 is used to multiplex the output of a DAC, breaking the single DAC output into four multiplexed channels. S&H circuits can be used to selectively delay the DAC's output until after the glitch, thereby smoothing the DAC output.

Multiple S&H circuits can be used to increase the throughput of an ADC by pipelining multiplexed inputs. Here, multiple S&Hs are connected in common to the multiplexer output. The ADC is connected to one S&H, which holds the input level for a conversion. The other S&Hs acquire other multiplexer channels, and in turn, connect to the ADC, while the first S&H is free to connect to another multiplexed channel. This pipelining technique eliminates the S&H acquisition time in the ADC signal path.

Many ADCs incorporate S&H or T&H circuits within their integrated package. An example is the Texas Instruments ADC121S021CIMFX, a 12-bit successive approximation register (SAR) ADC with built-in T&H that operates with sample rates in the range of 50 to 200 kilosamples per second (kS/s). It features a high-speed serial output bus which simplifies wiring layout (Figure 8).

This ADC is typical of many integrated ADC circuits in that it has an internal T&H, which simplifies pc board layout and helps minimize component count. External T&H circuits are used for special configurations such as differential input connections, multiplexed inputs, or when the ADC does not have an internal T&H or S&H circuit.

Conclusion

From audio recording to the most advanced IIoT or AI analysis, the most basic electronic function of converting an analog signal to a digital signal requires careful attention to S&H or T&H circuits. These are essential for minimizing voltage skew errors during the analog-to-digital conversion process as they hold the input voltage to the ADC constant during the conversion. The S&H can be internal to the ADC or external, but it must be in the signal path between the anti-aliasing low-pass filter and the ADC. As shown, there are many configurations—single, differential, or multiple devices per IC—to meet a variety of design applications. These applications extend to include preventing output transients or glitches caused by code transitions in DACs.

Recommended reading

1. Match the Right ADC to the Right Application
2. Sometimes the More Expensive ADC is the Most Cost Effective
3. Pay Attention to the ADC Precision Voltage Reference for Accurate IoT Sensing

The basics of low and high-pass filters

By Aiden Warne

Introduction to low and high-pass filters

Low and high-pass filters are used widely in the electrical engineering world. For example, these low and high-pass filters can be used in audio processing, image processing, communication systems, and sometimes biomedical signal processing. These filters do as their name suggests. They can filter out either a high frequency and let low frequencies pass, and they can do the opposite, filter out low frequencies and let high frequencies pass. In this blog, I am going to find the waveforms of the given high-pass and low-pass filter circuits for the components I am using. Upon building the circuit, I am going to compare the waveforms of the signal from the frequency to the waveform from the circuit components and see if they are the same or out of phase. I am going to use a function generator as well as an oscilloscope to measure these waveforms.

What are low and high-pass filters?

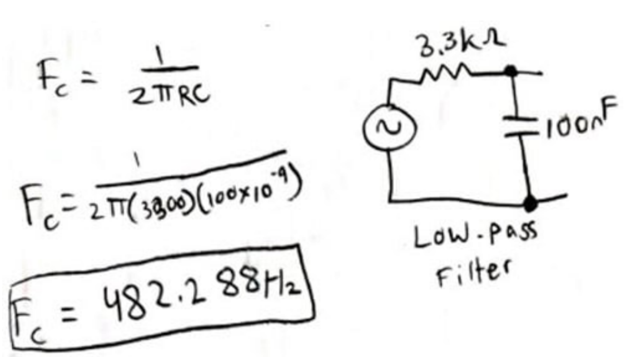
Low and high-pass filters, as discussed before, either filter out frequencies lower than the cut-off frequency or higher than the cut-off frequency. This cut-off frequency is determined by the equation listed below, where R is the resistance of the resistor, C is the capacitance of the capacitor, and 2π is a constant. This cut-off frequency and the order of the components determine how the circuit will respond to a given input frequency. A low-pass filter has the resistor first and then the capacitor second. Rather, the high-pass filter has the capacitor and then the resistor. If the input frequency is greater than the cut-off frequency and it is a high-pass filter, then the waveform will pass through unfiltered. If the input frequency is less than the cut-off frequency, then the waveform will be filtered and be out of phase and have a lower amplitude compared to the input waveform. The opposite is true when it comes to low-pass filters.

How to calculate frequency cut-off

As stated above, the cut-off frequency is the frequency in which the low or high-pass filter filters out the frequency, which is calculated in the equation: 12RC. As stated above, R is the resistance of your component and C is the capacitance of your component. All that is needed for a high-pass filter is one capacitor, one resistor, and of course some jumper wires and a power supply. For this project, I used a 3.3k resistor and a 100 nano farad capacitor. For low and high-pass filters, you can really use any component values you want to. I used these specific components for ease of use and to get nicer values for a cut-off frequency. As shown below, I put my values in the given equation and came up with a cut-off frequency of 482.288Hz. This is the frequency in which the circuit will balance between to make the high and low-pass filters.

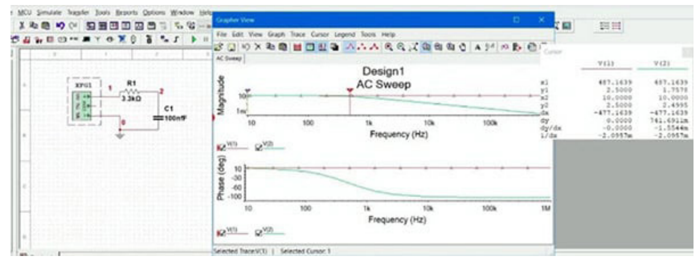
You can also use DigiKey's Low Pass/High Pass Filter Calculator to find these values.

Hand calculations



Simulations

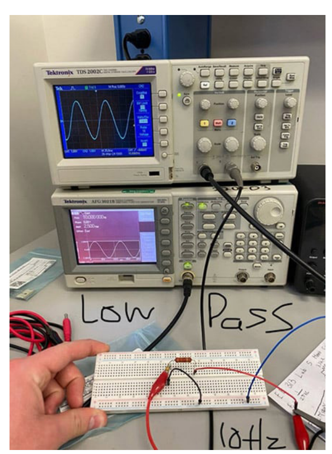
Multisim low-pass circuit



Here, I used a circuit simulation software called Multisim. With this, I was able to confirm my hand calculations by seeing that the higher frequency was indeed cut off after it reached the cut-off frequency of 482.288Hz. The graph shows that the amplitude is high in the lower frequency region, but as it crosses 482Hz, it starts to taper off and go in a negative direction. With the resistor first in order, in series with the capacitor, this creates the low-pass filter. The power source that is connected to it is a function generator, giving off a pulse that allows the waves to be seen on the oscilloscope. You can also do this with the high pass filter.

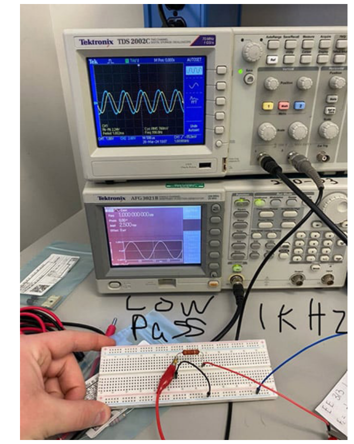
Physical circuit build

Low-pass 10Hz 3.3k Ohms 100nF



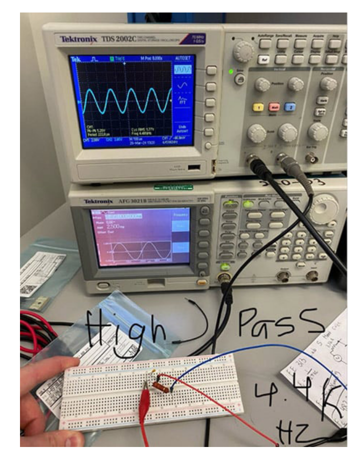
In this picture, the low-pass filter circuit is built on the breadboard. The waveform is applied first through the resistor and then through the capacitor. The input frequency applied is less than the cut-off frequency, thus the filtered waveform is in phase and has no difference in amplitude compared to the input waveform. According to my understanding of low-pass filters and comparing this to my hand calculations, this makes sense.

Low-pass 1kHz 3.3k Ohms 100nF



Again, the low-pass filter circuit is pictured above. This time, the input frequency applied is more than the cut-off frequency, thus the filtered waveform isn't in phase and there is a difference in amplitude compared to the input waveform. For reference, the blue waveform (the applied waveform) is at 2 volts per division and the yellow waveform (the filtered signal) is at 1 volt per division. According to my understanding of low-pass filters and comparing this to my hand calculations, this makes sense.

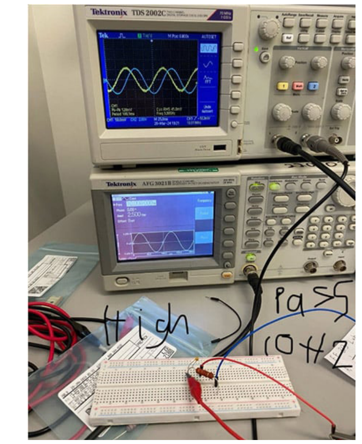
High-pass 4.4kHz 3.3k Ohms 100nF



In this picture, the high-pass filter circuit is built on the breadboard. The waveform is applied first through the capacitor and then through the resistor. This is the opposite of the low-pass filter. The input frequency is more than the cut-off frequency; thus, it will be in phase and has no difference in amplitude compared to the input waveform. According to my understanding of high-pass filters and comparing this to my hand calculations, this makes sense.

High-pass 10Hz 3.3k Ohms 100nF

Again, the high-pass filter circuit is pictured in the next column. This time the input frequency applied is less than the cut-off frequency, thus the filtered waveform isn't in phase and there is a difference in amplitude compared to the input waveform. For reference, the blue waveform (the applied waveform) is at 2 volts per division and the yellow waveform (the filtered signal) is at 50 millivolts per division. According to my understanding of low-pass filters and comparing this to my hand calculations, this makes sense.



Results

When doing the simulation and real experiments, I got very similar waveforms and they compared and matched what they should be. The low-pass filter, in which the cut-off was 482 Hz, allowed a 10 Hz frequency to go through, but not a 1K Hz to go through, which is what I was expecting. The high-pass filter had the expected outcome as well, with the high frequency waves lining up, and the lower ones not getting passed through.

Conclusion

In conclusion, I was able to get a better understanding of how low-pass and high-pass filters work, as well as how the frequency cut-off works with the given components. I was also able to clearly see the similarities and differences between the waveforms when the components were flipped around. You can make a high or low-pass filter with any combination of resistor and capacitor and can figure out the cut-off frequency the circuit will have using the equation I have provided. This can be a fun little project anyone can do with a few supplies and is a good little learning experience to anyone wanting to know more about frequencies in electronics.

Signal processing: Exponentially moving average (EMA) filter

By Mustahsin Zarif

Previously, in Introduction to Signal Processing, we've seen the two classes of filters: Finite Impulse Response (FIR) and Infinite Impulse Response (IIR). We saw how the moving average filter can be expressed in both a FIR and IIR form, but what are the benefits of one over the other?

Looking back at the example from my previous blog, the FIR filter expanded has the form:

y[5] = (x[5]+x[4]+x[3]+x[2]+x[1]+x[0])/5,

Here, we see that we require:

- 5 multiplication and
- 4 summation operations.

Multiplication operations are particularly computationally expensive. Therefore, if we look at the IIR form again, we see that it requires only:

- 3 multiplication and
- 2 summation operations

y[6]=(x[6]+y[5]-x[1])/5

This reduces the computation cost significantly! This is good for embedded devices such as microcontrollers since they spend less resources at every discrete time step to perform calculations.

As an example, when I use the Python function 'time.time' for the 11 point moving average filter in FIR and IIR form, with all the parameters (window size, sample rate, sample size, etc.) the same, I get the following runtime results respectively: 51 ms, 27 ms.

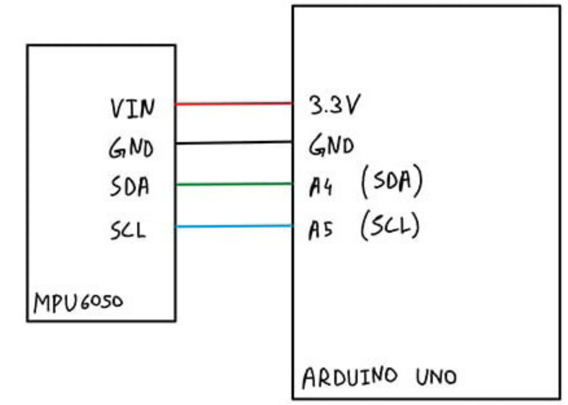


Figure 1: Block diagram connection between the MPU6050 and the Arduino Uno. (Image source: Mustahsin Zarif)

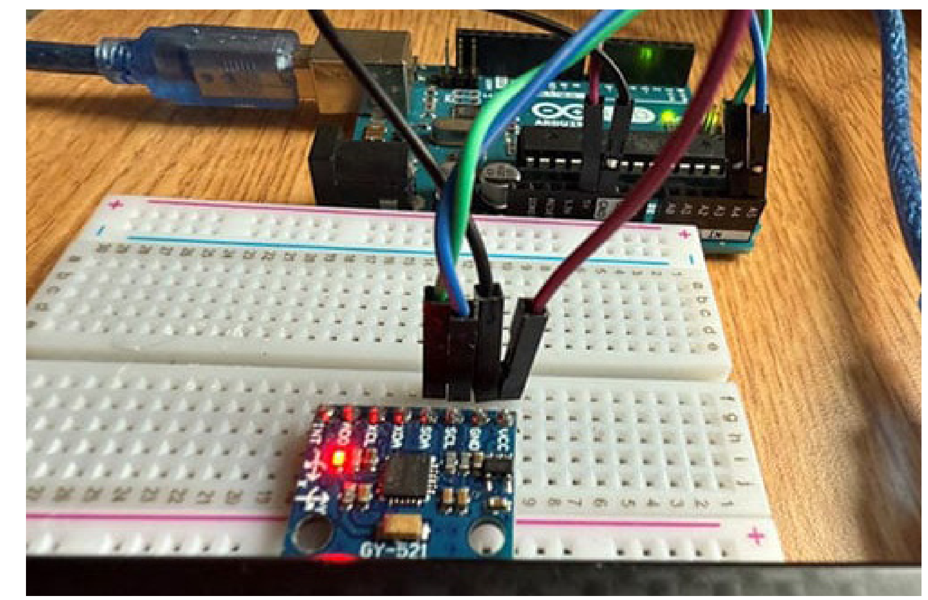


Figure 2: Connection between the MPU6050 and the Arduino Uno. (Image source: Mustahsin Zarif)

Discrete time IIR filter example

Now that we have an intuition as to why IIR filters perform better on microcontrollers, let's see an example project using an Arduino UNO and a DFRobot MPU6050 inertial measurement unit (IMU) (Figure 1). We will apply the exponentially moving average (EMA) filter to the IMU data to see differences between raw and smoothed data.

The exponentially moving average filter is of the recursive form:

$y[n] = \alpha*x[n] + (1- \alpha)*y[n-1]$

It is recursive because any current output that we are measuring depends on the previous outputs as well; i.e., the system has memory.

The constant alpha (α) determines how much weight we want to give to the current input as opposed to the previous outputs. For clarity, lets expand the equation to get:

$$y[n] = \alpha*x[n] + (1- \alpha)*(\alpha*x[n-1]+(1-\alpha)*y[n-2])$$

$$y[n] = \alpha*x[n] + (1- \alpha)*x[n-1]+\alpha*(1-\alpha)2*x[n-2])+ \ldots$$

$$y[n] = k=0n\alpha*(1-\alpha)k*x[n-k]$$

We see that the greater the alpha, the more the current input affects the current output. This is good since if the system is evolving, values far in the past are not as representative of the current system. On the other hand, this would be bad if, for example, there is a sudden, momentary change in the system from normal; in this case, we would like our output to follow the trend that the previous outputs were following.

Now without further ado, let's see how the code for an EMA filter would work for the MPU6050.

EMA Filter Code

When we run this code and check the serial plotter, we can see rough and smooth lines in pairs for accelerations in the x, y, and z axes, using a window size of 11 and an alpha value of 0.2 (Figure 3 to 5).

EMA Filter Code:

```
#include <wire.h>
#include <mpu6050.h>

MPU6050 mpu;

#define BUFFER_SIZE 11  // Window size

float accelXBuffer[BUFFER_SIZE];
float accelYBuffer[BUFFER_SIZE];
float accelZBuffer[BUFFER_SIZE];
int bufferCount = 0;

void setup() {
  Serial.begin(115200);
  Wire.begin();

  mpu.initialize();

  if (!mpu.testConnection()) {
    Serial.println("MPU6050 connection failed!");
    while (1);
  }

  int16_t ax, ay, az;
  for (int i = 0; i < BUFFER_SIZE; i++) {
    mpu.getMotion6(&ax, &ay, &az, NULL, NULL, NULL);
    accelXBuffer[i] = ax / 16384.0;
    accelYBuffer[i] = ay / 16384.0;
    accelZBuffer[i] = az / 16384.0;
  }
  bufferCount = BUFFER_SIZE;
}

void loop() {
  int16_t accelX, accelY, accelZ;

  mpu.getMotion6(&accelX, &accelY, &accelZ, NULL, NULL, NULL);

  float accelX_float = accelX / 16384.0;
  float accelY_float = accelY / 16384.0;
  float accelZ_float = accelZ / 16384.0;

  if (bufferCount < BUFFER_SIZE) {
    accelXBuffer[bufferCount] = accelX_float;
    accelYBuffer[bufferCount] = accelY_float;
    accelZBuffer[bufferCount] = accelZ_float;
    bufferCount++;
  } else {
    for (int i = 1; i < BUFFER_SIZE; i++) {
      accelXBuffer[i - 1] = accelXBuffer[i];
      accelYBuffer[i - 1] = accelYBuffer[i];
      accelZBuffer[i - 1] = accelZBuffer[i];
    }
    accelXBuffer[BUFFER_SIZE - 1] = accelX_float;
    accelYBuffer[BUFFER_SIZE - 1] = accelY_float;
    accelZBuffer[BUFFER_SIZE - 1] = accelZ_float;
  }

//calculate EMA using acceleration values stored in buffer
  float emaAccelX = accelXBuffer[0];
  float emaAccelY = accelYBuffer[0];
  float emaAccelZ = accelZBuffer[0];
  float alpha = 0.2;

  for (int i = 1; i < bufferCount; i++) {
    emaAccelX = alpha * accelXBuffer[i] + (1 - alpha) * emaAccelX;
    emaAccelY = alpha * accelYBuffer[i] + (1 - alpha) * emaAccelY;
    emaAccelZ = alpha * accelZBuffer[i] + (1 - alpha) * emaAccelZ;
  }

  Serial.print(accelX_float); Serial.print(",");
  Serial.print(accelY_float); Serial.print(",");
  Serial.print(accelZ_float); Serial.print(",");
  Serial.print(emaAccelX); Serial.print(",");
  Serial.print(emaAccelY); Serial.print(",");
  Serial.println(emaAccelZ);

  delay(100);
}
</mpu6050.h></wire.h>
```

Figure 3: Raw and filtered acceleration values in the x-direction. (Image source Mustahsin Zarif)

Figure 4: Raw and filtered acceleration values in the y-direction. (Image source Mustahsin Zarif)

Figure 5: Raw and filtered acceleration values in the z-direction. (Image source Mustahsin Zarif)

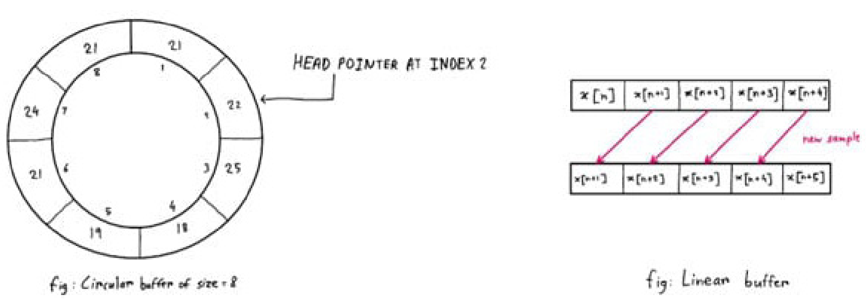


Figure 6: Example illustration of a circular buffer. (Image source: Mustahsin Zarif)

Making the code one step smarter

We now have an idea of how IIR filters are better for controllers compared to FIR filters because of the significantly fewer summation and multiplication calculations required. However, when we implement this code, summation and multiplication aren't the only calculations being performed: we have to shift the samples every time a new time sample comes in, and this process, under the hood, requires compute power. Therefore, instead of shifting all of the samples at every sampling time interval, we can employ the help of circular buffers.

Here is what we do: we have a pointer that remembers the index of the data sample that came in. Then, every time the pointer points to the last element in the buffer, it points to the first element of the buffer next, and the new data replaces the data that was stored here before, since this is now the oldest data that we do not need anymore (Figure 6). Consequently, this method allows us to keep track of the oldest sample in the buffer and replace that one without having to shift samples every time to put the new data in the last element of the array.

This is what the code looks like for an EMA filter implementation using circular buffers. Can you try to run this for a gyroscope instead of an accelerometer? Play around with the coefficients too!

Summary

In this blog, we discussed the difference between IIR and FIR filters with an emphasis on their computational efficiencies. By taking a small example of the reduction in number of operations required from FIR to IIR, we can imagine how efficient IIR filters will be when applications are scaled, which is important for real-time applications on limited hardware power.

We also took a look at an example project using an Arduino Uno and MPU6050 IMU, where we deployed an exponentially moving average filter to reduce noise in sensor data while still capturing the underlying signal behavior. Finally, in the interest of efficiency, we also saw an example of smarter code by employing circular buffers instead of shifting data at every time interval.

In the next blog, we will use Red Pitaya's FPGA functionality to implement a 4 tap FIR filter digital circuit!

EMA Filter Using a Circular Buffer Code:

```
#include <wire.h>

#include <mpu6050.h>

MPU6050 mpu;

#define BUFFER_SIZE 11  // Window size

float accelXBuffer[BUFFER_SIZE];

float accelYBuffer[BUFFER_SIZE];

float accelZBuffer[BUFFER_SIZE];

int bufferIndex = 0;

void setup() {

  Serial.begin(115200);

  Wire.begin();


  mpu.initialize();


  if (!mpu.testConnection()) {

    Serial.println("MPU6050 connection failed!");

    while (1);

  }

  int16_t ax, ay, az;

  for (int i = 0; i < BUFFER_SIZE; i++) {

    mpu.getMotion6(&ax, &ay, &az, NULL, NULL, NULL);

    accelXBuffer[i] = ax / 16384.0;

    accelYBuffer[i] = ay / 16384.0;
    accelZBuffer[i] = az / 16384.0;

  }

}

void loop() {

  int16_t accelX, accelY, accelZ;

  mpu.getMotion6(&accelX, &accelY, &accelZ, NULL, NULL, NULL);

  float accelX_float = accelX / 16384.0;

  float accelY_float = accelY / 16384.0;

  float accelZ_float = accelZ / 16384.0;

  accelXBuffer[bufferIndex] = accelX_float;

  accelYBuffer[bufferIndex] = accelY_float;

  accelZBuffer[bufferIndex] = accelZ_float;

  bufferIndex = (bufferIndex + 1) % BUFFER_SIZE; //circular buffer implementation

  float emaAccelX = accelXBuffer[bufferIndex];

  float emaAccelY = accelYBuffer[bufferIndex];

  float emaAccelZ = accelZBuffer[bufferIndex];

  float alpha = 0.2;

  for (int i = 1; i < BUFFER_SIZE; i++) {

    int index = (bufferIndex + i) % BUFFER_SIZE;

    emaAccelX = alpha  accelXBuffer[index] + (1 - alpha)  emaAccelX;

    emaAccelY = alpha  accelYBuffer[index] + (1 - alpha)  emaAccelY;

    emaAccelZ = alpha  accelZBuffer[index] + (1 - alpha)  emaAccelZ;
  }

  Serial.print(accelX_float); Serial.print(",");

  Serial.print(emaAccelX); Serial.print(",");

  Serial.print(accelY_float); Serial.print(",");

  Serial.print(emaAccelY); Serial.print(",");

  Serial.print(accelZ_float); Serial.print(",");

  Serial.println(emaAccelZ);

  delay(100);

}
</mpu6050.h></wire.h>
```

Improve test performance with low-cost signal sources using inline filters

By Art Pini
Contributed By DigiKey's North American Editors

Do you need a pure sine wave for a test but only have an arbitrary function generator with a high harmonic level available? Perhaps you're mixing the outputs of two signal generators and must select the upper sideband component at the mixer output. How can you do this? The solution is to use inline RF filters, such as Crystek Corporation's CLPFL-0200 (Figure 1, left) with an SMA connector and CLPFL-0021-BNC (Figure 1) with a BNC connector.

Figure 1: Inline coaxial filters, such as the CLPFL-0200 with an SMA connector (left) or the CLPFL-0021 with a BNC connector (right), can reduce signal harmonics and noise on signal sources. (Image source: Crystek Corporation)

RF filters clean up signals by selectively attenuating unwanted frequencies while passing desired frequency components. Inline filters, intended for use with coaxial lines, are designed with a 50 ohm (Ω) nominal impedance. These filters reduce noise by reducing the signal bandwidth. They also control the signal spectrum to reduce harmonics, images, and interfering signals.

Types of filters

There are several types of inline filter configurations, including low pass, high pass, and bandpass (Figure 2).

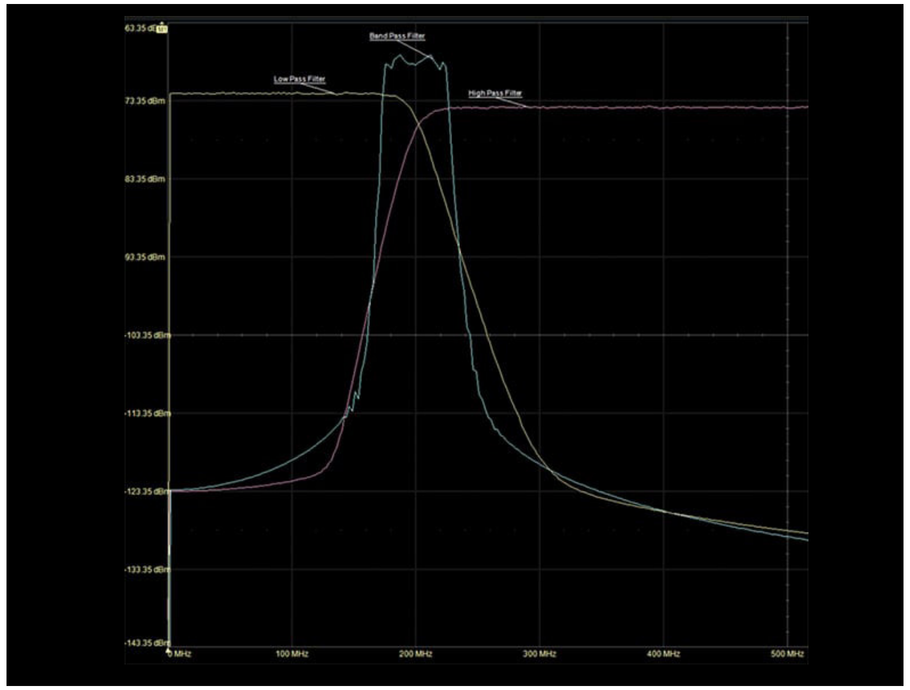


Figure 2: Shown are the frequency responses of low-pass, high-pass, and bandpass filters. (Image source: Art Pini)

Low-pass filters pass frequencies below a fixed cutoff and can eliminate the harmonics of a signal with the cutoff set just above the fundamental frequency. High-pass filters pass frequencies above a fixed cutoff and can eliminate an interfering signal with the cutoff set above the power line frequency. Bandpass filters attenuate unwanted signals by passing frequencies within a desired band and can be employed as a preselector for an RF front-end. The region where the signal is transmitted with little loss is called the passband, and the region where the signal is highly attenuated is the stopband. The region(s) between the passband and the stop band is the transition region(s).

Selecting the right filter

Filters are designed for specific frequency response characteristics. These include the sharpness of the transition from the passband to the stopband, the flatness of the passband and the stopband, and the phase response as a function of frequency. There are several classic designs shown in Figure 3.

The Butterworth filter has a flat passband response and a moderate roll-off rate. The Bessel filter has the most linear phase response but the slowest roll off; it would typically be used when a band-limited pulse waveform must be transmitted with minimum distortion. The Chebyshev filter has a fast roll off but has ripple in the passband. The inverse Chebyshev filter has a flat passband response and a fast roll off but exhibits ripple in the stop band. The Butterworth and the Chebyshev are two of the most widely used inline filters.

The roll-off characteristics of any filter type are affected by its order. The order is derived from the filter's transfer function and indicates the number of poles in the design. In general, the higher the filter order, the faster the roll off (Figure 4).

Crystek's CLPFL-0200 is a 7th-order Butterworth low-pass filter with a passband of DC to 200 megahertz (MHz) and an insertion loss of 2.2 decibels (dB) at a

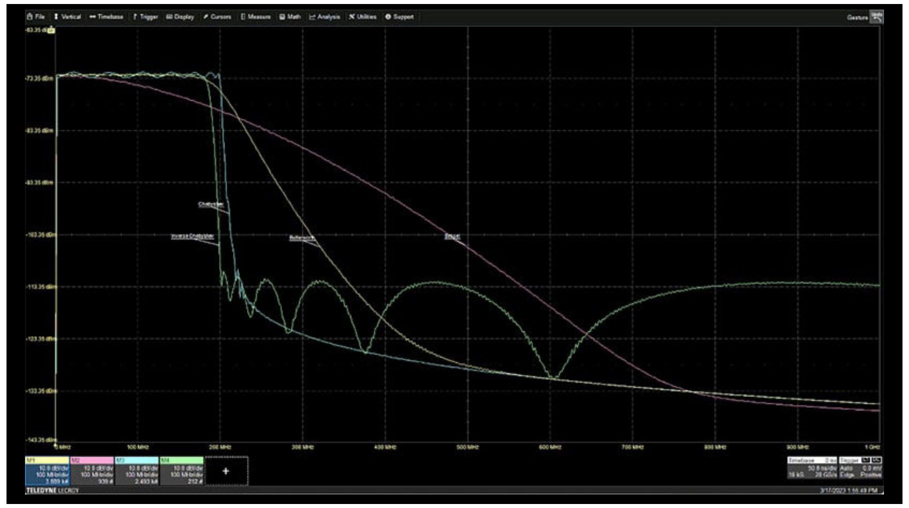


Figure 3: The frequency response of several types of classic filters shows the differences in roll-off and flatness characteristics. (Image source: Art Pini)

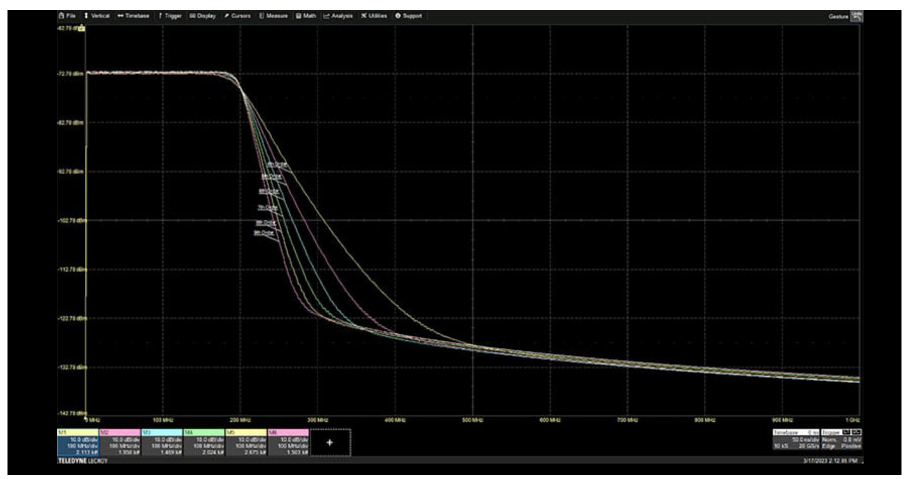


Figure 4: Shown is a comparison of a Butterworth low-pass filter response for a filter with orders of 5 through 9. The higher the filter order, the faster the roll off in the transition region. (Image source: Art Pini)

frequency of 210 MHz. This filter could be used to clean up the output of a signal generator when making an effective number of bits (ENOB) measurement on an 8-bit analog-to-digital converter (ADC) (Figure 5).

The upper trace shows the signal generator output spectrum with a second harmonic only 22 dB below the fundamental. With the filter (lower trace), the second harmonic is down over 70 dB, and other harmonics are below the noise floor.

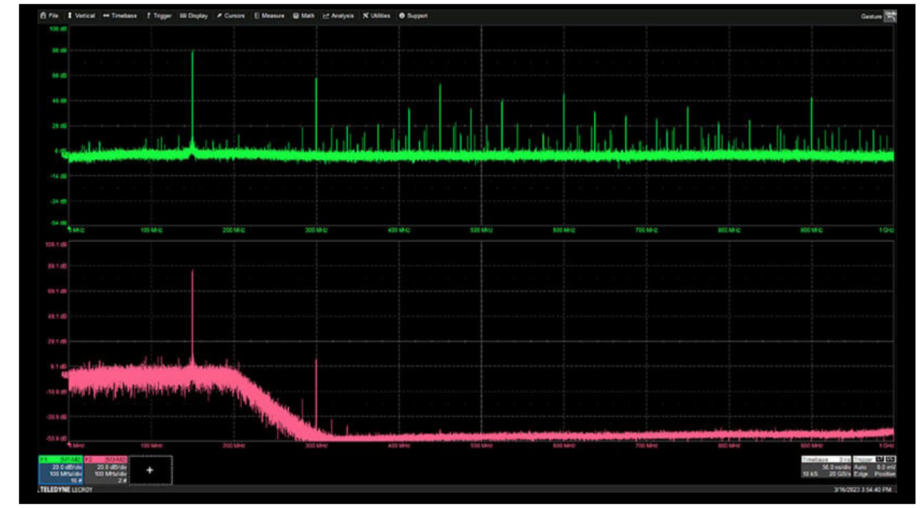


Figure 5: Shown is the result of a 200 MHz low-pass filter being used to remove harmonics and noise from a signal generator. The filtered signal (lower trace) has significantly reduced noise and harmonic levels. (Image source: Art Pini)

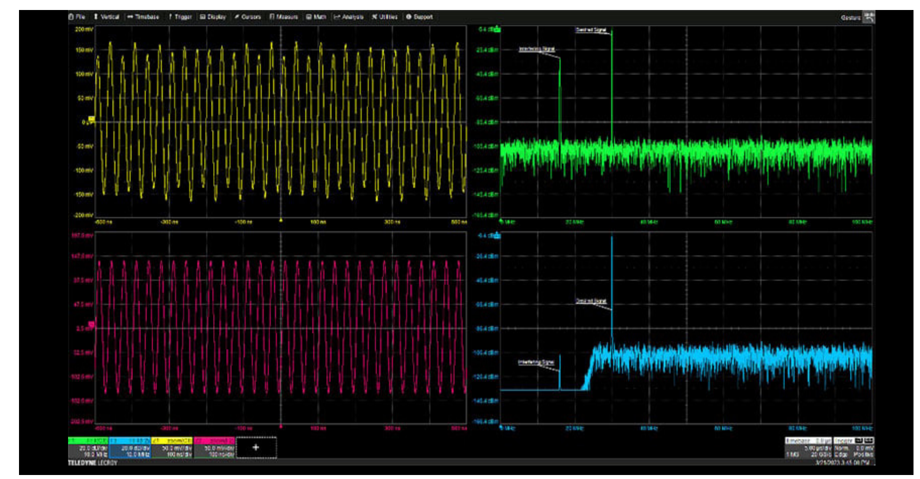


Figure 6: Shown is a high-pass filter being used to eliminate a 13 MHz interfering signal from the desired 30 MHz signal (upper trace). The filtered signal appears in the lower trace. (Image source: Art Pini)

Note also that the noise floor above the filter cutoff frequency is lowered by better than 40 dB.

High-pass filters eliminate interfering signals with a lower frequency than the desired signal (Figure 6).

In Figure 6, a high-pass filter attenuates a 13 MHz interfering signal and passes the 30 MHz signal of interest. The interfering signal's effect can be seen in the time-domain view (upper left) as an amplitude variation of the signal peaks. The filtered signal (lower left) has flat peak amplitudes.

A filter such as Crystek's CHPFL-0025-BNC, a 7th-order 25 MHz Chebyshev high-pass filter with BNC connectors, could attenuate the interfering signal.

Crystek filters are offered in up to 9th-order configurations. For example, the CLPFL-0021-BNC mentioned earlier is a 21 MHz Chebyshev response, 9th-order, low-pass filter. It delivers a transition region that rolls off at about 55 dB per octave.

Bandpass filters typically require more components than low or high-pass filters, which take up space and add to the BOM. Crystek addresses this using surface acoustic wave (SAW) technology to allow its bandpass filters to fit in the same package as low-pass or high-pass filters. An example SAW bandpass filter is the Crystek CBPFS-0915 with SMA connectors and a 26 MHz bandwidth centered on 915 MHz.

Conclusion

Inline RF filters improve test performance by eliminating harmonics, noise, and interference from signal sources. Companies like Crystek offer a wide range of inline filters to match your signal-conditioning needs.

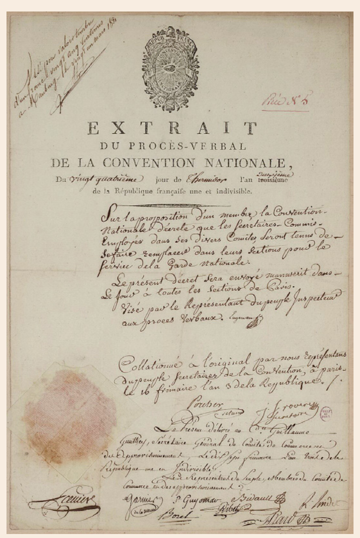
This month in history

1795

April 7

Adoption of the metric system

In the midst of the French Revolution, France's National Convention formally adopted the metric system in law. A new standard of weights and measures that later spread worldwide and became essential to science and engineering.


EXTRAIT
DU PROCÈS-VERBAL
DE LA CONVENTION NATIONALE,

The National Convention's decree of 18 Germinal Year III legally established a new decimal system of measurement in France as standard units and banned the old royal measures.

1820

April 21

Ørsted's electromagnetism discovery

Danish physicist Hans Christian Ørsted noticed that a compass needle deflected when an electric current was switched on. This accidental discovery in a classroom launched the age of electromagnetism, revealing that electricity and magnetism are linked and directly inspiring the work by Ampère, Ohm, Faraday, and Maxwell.

Hans Christian Ørsted (1777–1851), shown in a period portrait, was the first to detect that electricity and magnetism are intertwined. This news spread through Europe like fire.

1839

April 9

First commercial electric telegraph

The world's first commercial telegraph line ran 13 miles from London to West Drayton. Within a few years, telegraph networks were connecting cities and continents, shrinking communication times from days or weeks to minutes.

The initial telegraph line ran alongside the tracks from Paddington Station in London to West Drayton. Railway signalmen quickly adopted the telegraph to dispatch trains more safely.

1939

April 30

TV debuts at New York World's Fair

The New York World's Fair opened, and that same day, RCA initiated regular public television broadcasts in the U.S. NBC transmitted President Roosevelt's Fair opening address to a few hundred TV sets in the New York area, the first U.S. president on TV. Thousands of fairgoers also saw televised images at RCA's pavilion.

The curious gather around a television demonstration at the RCA exhibit during the 1939 fair. For many, this was the first time seeing moving images transmitted through the air.

1954

April 25

First solar battery demonstrated

Bell Laboratories scientists unveiled the first practical silicon solar cell, converting sunlight to electricity at about 6% efficiency. In a public demonstration, a solar panel powered a small toy Ferris wheel, in an event that launched the field of photovoltaics. This "solar battery" was soon used to run telephone equipment and to power the Vanguard satellite, seeding today's solar power industry.

Bell Labs researchers Gerald Pearson, Daryl Chapin, and Calvin Fuller demonstrate their new silicon "solar battery."

1973

April 3

First handheld mobile phone call

Martin Cooper of Motorola made the world's first public cell phone call on April 3, 1973, using a prototype DynaTAC handset. Standing on a New York City sidewalk, he dialed a rival at Bell Labs and exclaimed, "Hi, I'm calling you from a cell phone!"

Martin Cooper holding the prototype DynaTAC cellphone on the streets of NYC in 1973.


The parts we sell help turn inspiration into reality
From millions of parts to cutting-edge productivity tools, DigiKey helps turn your innovative ideas into reality, regardless of where inspiration strikes.
Discover millions of parts at digikey.com or call 1.800.344.4539
DigiKey
we get technical
DigiKey is an authorized distributor for all supplier partners. New products added daily. DigiKey and DigiKey Electronics are registered trademarks of DigiKey Electronics in the U.S. and other countries. © 2026 DigiKey Electronics, 701 Brooks Ave. South, Thief River Falls, MN 56701, USA
ECIA MEMBER
Supporting The Authorized Channel