Electrical, Computer & Energy Engineering
UNIVERSITY OF COLORADO BOULDER

# ECEN 5053-003 Homework Assignment

Course Name: Embedding Sensors and Actuators
Corresponding Module: C1M4
Week Number: 4
Module Name: Amplifiers and Sensor Noise
Student Name: Rushi James Macwan

Note: Correct answer is in Blue Font

Homework is worth 100 points for Parts A and B combined.

Part 1: Each question is worth 5 points.

A. An inverting op amp has $V_{in}$ = .056 volts, $R_2$ = 1000 ohms and $R_1$ = 50 ohms. What is $V_{out}$ in volts?

Sol. **-1.12 Volts**

Based on the class slide, the inverting amplifier will use the below given formula:

$(V_{in}/R_1) + (V_{out}/R_2) = 0$ … [1]

Using the above presented formula, we can calculate the output voltage $V_{out}$ in Volts.

Thus, $(0.056/50) + (V_{out}/1000) = 0$

This leads to $0.00112 + (0.001*Vout) = 0$

Based on this development, we get the output voltage which is: $V_{out}$ = -0.00112/0.001 = **-1.12 Volts**

Courtesy: To solve this problem, I have used the class slide: C1M4V3.

B. A non-inverting op amp has $V_{out}$ = 2.55 volts, $R_2$ = 2000 ohms and $R_1$ = 100 ohms. What is $V_{in}$ in volts?

Sol. **0.1214 Volts**

Based on the class slide, the non-inverting amplifier will use the below given formula:

$(V_{out}/(R_1 + R_2)) - (V_{in}/R_1) = 0$ … [1]

Using the above presented formula, we can calculate the input voltage $V_{in}$ in Volts.

Thus, $(V_{in}/R_1) = (V_{out}/(R_1 + R_2))$

This leads to $V_{in}$ = R1 * $(V_{out}/(R_1 + R_2))$ = 100 * (2.55/(2000+100)) = 0.1214 Volts

Based on this development, we get the input voltage which is: $V_{in}$ = **0.1214 Volts**

Courtesy: To solve this problem, I have used the class slide: C1M4V3.

C. An A summing amplifier with N = 3 has $R_F$ = 2500 ohms, $V_1$ = 0.154 volts, $R_1$ = 200 ohms; $V_2$ = .059 volts and $R_2$ = 100 ohms; $V_3$ = .104 volts and $R_3$ = 500 ohms. What is $V_{out}$ in volts?

Sol. **-3.92 Volts**

Based on the class slide, the summing amplifier will use the below given formula (as in our case):

$V_{out}$ = - (RFV1/R1 + RFV2/R2 + RFV3/R3)

Using the above presented formula, we can calculate the output voltage $V_{out}$ in Volts.

Thus, $V_{out}$ = - [2500 * ((0.154/200) + (0.059/100) + (0.104/500))]

This leads to $V_{out}$ = - 3.92 Volts

Based on this development, we get the output voltage which is: $V_{out}$ = **-3.92 Volts**

Courtesy: To solve this problem, I have used the class slide: C1M4V3.

D. A differential amplifier has R4 = 5000 ohms, V1 = 0.294 volts, R3 = 500 ohms and V2 = -.094 volts. What is $V_{out}$ in volts?

Sol. **3.88 Volts**

Based on the class slide, the differential amplifier will use the below given formula (based on the superposition principle):

$V_{out} = (V_1 – V_2) (R_4 / R_3)$ … [1]

Using the above presented formula, we can calculate the output voltage $V_{out}$ in Volts.

Thus, $V_{out}$ = (0.294 - (- 0.094)) x (5000 / 500) = **3.88 V**

Courtesy: To solve this problem, I have used the class slide: C1M4V3.

E. You are using an instrumentation amplifier to measure the difference between two very small sensor signals. $V_1$ = 41.27 mV, $V_2$ = 38.43 mV, $R_4$ = 5000 ohms, $R_3$ = 100 ohms, $R_2$ = 3000 ohms and $R_1$ = 500 ohms. What is $V_{out}$ in milli-volts?

Sol. $\boxed{\mathbf{-1846\ mV}}$

Based on the class slide, the instrumentation amplifier will use the below given formula (based on the superposition principle):

Vout = (V2 – V1) x (R4/R3) x (R1+2R2)/R1) … [1]

Using the above presented formula, we can calculate the output voltage $V_{out}$ in Volts.

Thus, $V_{out}$ = (38.43m – 41.27m) x (5000/100) x ((500 + (2 x 3000)) / 500) =

Therefore $V_{out}$ = (-2.84m) x (50) x (6500 / 500) = -1.846 Volts

Based on this development, we get the output voltage which is:
$V_{out}$ = $\boxed{\mathbf{-1846\ mV}}$

Courtesy: To solve this problem, I have used the class slide: C1M4V4.

F. Suppose you have an amplifier with an input offset voltage of .08 mV as measured at 25 °C and a thermal drift of the input offset voltage of 0.4 μV/ºC away from this temperature. Your amplifier has a gain of 90, and the circuit in which it is installed in operating an elevated temperature of 40°C. How much is the output in mV offset by your input offset voltage?

Sol. $\boxed{\mathbf{7.74\ mV}}$

As given in the problem, the amplifier has an input offset voltage of 0.08 mV which is measured at 25°C and the amplifier has a thermal drift of the input offset voltage which is 0.4 μV/ºC. Based on this data, using the information provided in the class slide, the total input offset voltage at 40 °C can be calculated as below using the difference in temperature which is 15°C:

= Thermal drift at 40°C (which exists between 25°C and 40°C at a rate of 0.0004 mV/°C) + input offset voltage

= (0.0004 mV x 15 °C) + 0.08 mV = 0.086 mV

Based on the above calculation, the output in mV offset can be calculated by using the total input offset voltage as calculated above and the gain of the amplifier.

Thus, the output in mV offset = Gain x Total input offset voltage at 40 °C = 90 x (0.086 mV) = **7.74 mV**

Courtesy: To solve this problem, I have used the class slide: C1M4V5.

G. Suppose an op amp with a slew rate of 0.24 V/µs is amplifying the signal 6 sin (20,000,000t) mV. Will the amplified signal be distorted? Why or why not?

Sol. **No, the amplified signal will not be distorted.**

***The amplified signal will not be distorted because the minimum slew rate required for the given amplifying signal is 0.12 V/µs which is lesser than the slew rate provided in the problem which is 0.24 V/µs.***

To solve this problem, I have used the information provided in the class slide on slew rate distortion. Based on the information presented in the slide, an Op-Amp needs a minimum slew rate of $2\pi f V_0$ to amplify an input signal which is $V_0 \sin(2\pi ft)$ without distortion. Running the Op-Amp above the slew rate will distort the input signal and therefore we first calculate the minimum slew rate:

Based on the data provided in the question, the amplifying signal is 6 sin(2 x 10^(7) x t) mV which when compared with the equation $V_0 \sin(2\pi ft)$ provides the following value:

$V_0$ = 6 mV and $2\pi f$ = 2 x 10^(7) Hz

Thus, the minimum required slew rate is $2\pi f V_0$ = (6 mV) x (2 x 10^(7) Hz) = 12 x 10^(4) V/s = 0.12 V/µs

The given Op-Amp has however a slew rate of 0.24 V/µs which is higher than the minimum slew rate value of 0.12 V/µs and therefore the signal will not be distorted.

Courtesy: To solve this problem, I have used the class slide: C1M4V6.

H. The Johnson-Nyquist noise in a resistor in a circuit of 2 Mhz bandwidth is 60 nV / √hz. What is the magnitude of the noise in the resistor in μV?

Sol. $\boxed{\mathbf{84.853\ \mu V}}$

In this problem, the Johnson-Nyquist noise in the resistor in a given circuit of 2 MHz bandwidth is given as 60 nV / √hz. Therefore, to calculate the magnitude of the noise in the resistor in μV, we will need to calculate the Johnson-Nyquist noise for the entire bandwidth the noise given in the problem is for every √hz.

Thus, the magnitude of the noise in the resistor for the entire bandwidth is:

= (60 nV / √hz) x (√(2 x 10^(6)) ) = 84.853 x 10^(3) nV = **84.853 μV**

Courtesy: To solve this problem, I have used the class slide: C1M4V8.

I. What is the signal to quantization noise ratio (SNR) of a sine wave in a 16 bit ADC?

Sol. $\boxed{\mathbf{98.097\ dB}}$

To solve this problem, I have used the class slide on Quantization Noise Model as a reference. Based on the information provided in that slide, the sine wave quantization error is non-uniform.

So, SNR = 6.021Q + 1.761 dB [where, Q = bits in ADC = 16]
= 6.021 x 16 + 1.761 dB
= **98.097 dB**

Courtesy: To solve this problem, I have used the class slide: C1M4V9.

J. Your raw thermocouple sensor signal is 31 mV, and you are using an instrumentation amplifier to process it. The amplifier has a CMRR of 60 dB and a differential mode gain of 110 (assumed to be absolute value). If the RF noise on the leads from the thermocouple sensor to the data logger is 34 mV, what will be the percentage of noise on the amplified signal?

Sol. $\boxed{\mathbf{0.109677\ \%}}$

To solve this problem, I have used the common mode rejection formula that involves the use of both differential mode gain as well as common mode gain.

Thus, CMRR (dB) = 20 x $\log_{10}$ ($A_{diff}$/ $A_{comm}$) … [1]

Where, CMRR = Common Mode Rejection Ratio
$A_{diff}$ = Differential Mode Gain
$A_{comm}$ = Common Mode Gain

Using the equation [1] and substituting the values of CMRR (dB) and differential gain, we can obtain the value of common mode gain $A_{comm}$.

Thus, 60 (dB) = 20 x $\log_{10}$ (110 / $A_{comm}$)

∴ 3 = $\log_{10}$ (110 / $A_{comm}$)

∴ 10^3 = 110 / $A_{comm}$

∴ $A_{comm}$ = 0.110 (absolute value)

Based on the value obtained for common mode gain, we can find the RF noise output for the instrumentation amplifier as below:

= $A_{comm}$ x (RF noise on the leads from the thermocouple sensor to the data logger)

= 0.110 x 34 mV = 3.74 mV … [2]

Now, as for the thermocouple sensor signal which is 31 mV is applicable as a differential mode signal and therefore the output corresponding to this raw thermocouple sensor signal will be as follows:

= differential signal x differential gain
= 31 mV x 110
= 3410 mV … [3]

Now, to calculate the percentage of noise, we divide equation [2] by equation [3] where equation [2] is the noise output and equation [3] is the amplified sensor signal.

Thus, percentage of noise on the amplified signal:

= Eq [2] / Eq [3]
= (3.74 mV / 3410 mV) x 100 %
= **0.109677 %**

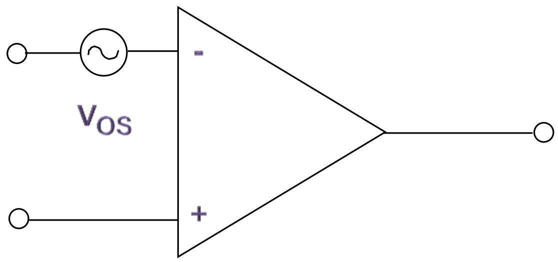
Courtesy: To solve this problem, I have used the class slide: C1M4V5.

Part 2: Each question is worth 10 points.

K. How does Analog Devices Inc. measure the input offset voltage on their amplifiers?

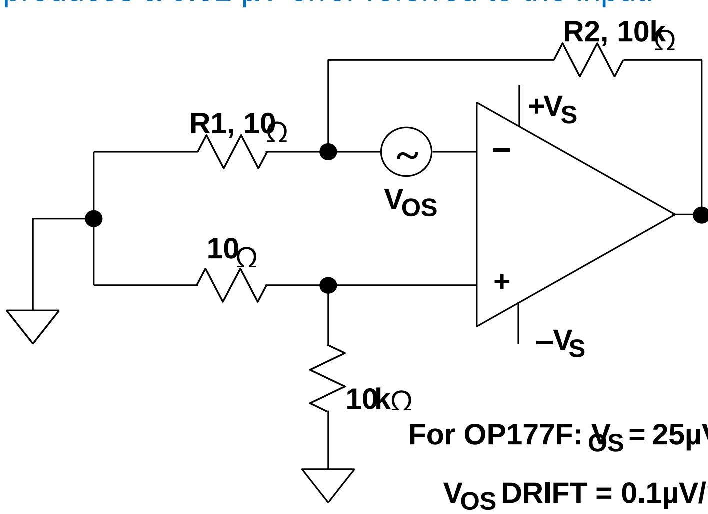
Sol.

Ideally, if both inputs of an op amp are at exactly the same voltage, then the output should be at zero volts. In practice, a small differential voltage must be applied to the inputs to force the output to zero. This is known as the *input offset voltage*, $V_{OS}$. Input offset voltage is modeled as a voltage source, $V_{OS}$, in series with the inverting input terminal of the op amp as shown in figure given below.




- **Offset Voltage: The differential voltage which must be applied to the input of an op amp to produce zero output.**

Measuring input offset voltages of a few microvolts requires that the test circuit does not introduce more error than the offset voltage itself. The below given figure shows a standard circuit for measuring offset voltage. The circuit amplifies the input offset voltage by the noise gain of 1001. The measurement is made at the amplifier output using an accurate digital voltmeter. The offset referred to the input (RTI) is calculated by dividing the output voltage by the noise gain. The small source resistance seen by the inputs results in negligible bias current contribution to the measured offset voltage. For example, 2 nA bias current flowing through the 10 Ω resistor produces a 0.02 µV error referred to the input.




$$V_{OUT} = \left[1 + \frac{R2}{R1}\right] V_{OS}$$

$$V_{OUT} = 1001\ V_{OS}$$

$$V_{OS} = \frac{V_{OUT}}{1001}$$

**For OP177F: $V_{OS}$ = 25µV max @ +25C**

**$V_{OS}$ DRIFT = 0.1µV/°C maximum**

**$V_{OS}$ STABILITY = 0.3µV/month typical**

***Figure: Measuring Input Offset Voltage***

Some characteristics and quirks of this measuring technique:

As simple as this circuit looks, it can give inaccurate results when testing precision op amps, unless care is taken in implementation. The largest potential error source comes from parasitic thermocouple junctions, formed where two different metals are joined. This thermocouple voltage can range from 2 µV/ºC to more than 40 µV/ºC. Note that in this circuit additional "dummy" resistors have been added to the non-inverting input, in order to exactly match/balance the thermocouple junctions in the inverting input path.

Accuracy of this measuring technique:

The accuracy of the measurement also depends on the mechanical layout of the components and exactly how they are placed on the PC board. Keep in mind that the two connections of a component such as a resistor create two equal, but opposite polarity thermoelectric voltages (assuming they are connected to the same metal, such as the copper trace on a PC board). These will cancel each other, *assuming both are at exactly the same temperature*. Clean connections and short lead lengths help to minimize temperature gradients and increase the accuracy of the measurement.

A few details about the test circuit:

In the test circuit, airflow should be minimal so that all the thermocouple junctions stabilize at the same temperature. In some cases, the circuit should be placed in a small closed container to eliminate the effects of external air currents. The circuit should be placed flat on a surface so that convection currents flow up and off the top of the board, not across the components, as would be the case if the board were mounted vertically.

Measuring the offset voltage shift over temperature is an even more demanding challenge. Placing the printed circuit board containing the amplifier being tested in a small box or plastic bag with foam insulation prevents the temperature chamber air current from causing thermal gradients across the parasitic thermocouples. If cold testing is required, a dry nitrogen purge is recommended. Localized temperature cycling of the amplifier itself using a Thermostream-type heater/cooler may be an alternative, however these units tend to generate quite a bit of airflow that can be troublesome. Generally, the test circuit of Figure 2 can be made to work for many amplifiers. Low absolute values for the small resistors (such as 10 Ω) will minimize bias current induced errors.

Courtesy: To solve this problem, I have used a reference which can be accessed by clicking the link: [1] [2] [3] [4] [5]

L. How does Analog Devices Inc. measure the common mode rejection ratio on their amplifiers?

Sol.

If a signal is applied equally to both inputs of an op amp, so that the differential input voltage is unaffected, the output should not be affected. In practice, changes in common mode voltage will produce changes in output. The op amp *common-mode rejection ratio* (CMRR) is the ratio of the common-mode gain to differential-mode gain. For example, if a differential input change of Y volts produces a change of 1 V at the output, and a common-mode change of X volts produces a similar change of 1 V, then the CMRR is X/Y. When the common-mode rejection ratio is expressed in dB, it is generally referred to as common-mode rejection (CMR)—*please note that there is very little consistency in this throughout the semiconductor industry with regards to the use of dB or ratio values for CMR or CMRR*.

Typical low frequency CMR values can be between 70 dB and 120 dB, but at higher frequencies, CMR deteriorates. In addition to a CMRR numeric specification, many op amp data sheets show a plot of CMR versus frequency, as shown in Figure 1 for the OP177 precision op amp.

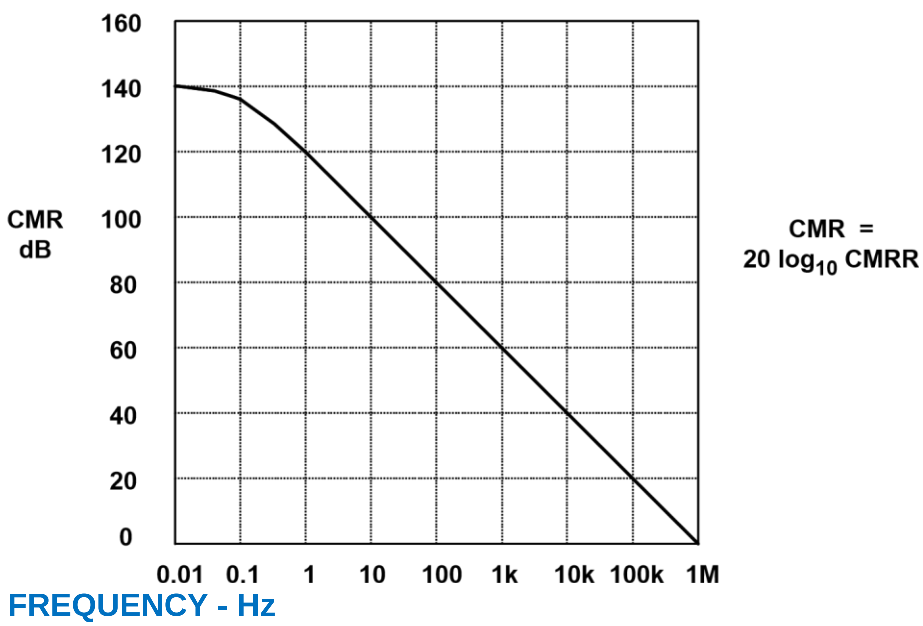



***Figure 1: CMRR for the OP177***

CMRR produces a corresponding output offset voltage error in op amps configured in the noninverting mode as shown in Figure 2. Note inverting mode operating op amps will have less CMRR error. Since both inputs are held at a ground (or virtual ground), there is no CM dynamic voltage.

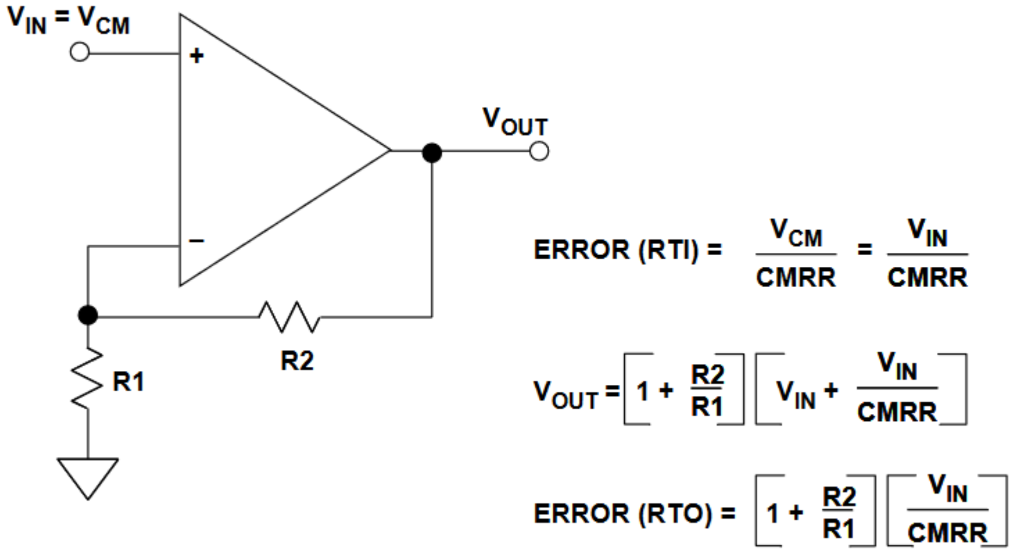



*Figure 2: Calculating Offset Error Due to Common-Mode Rejection Ratio (CMRR)*

## MEASURING COMMON MODE REJECTION RATIO

Common-mode rejection ratio can be measured in several ways. The method shown in Figure 3 below uses four precision resistors to configure the op amp as a differential amplifier, a signal is applied to both inputs, and the change in output is measured—an amplifier with infinite CMRR would have no change in output. The disadvantage inherent in this circuit is that the ratio match of the resistors is as important as the CMRR of the op amp. A mismatch of 0.1% between resistor pairs will result in a CMR of only 66 dB—no matter how good the op amp! Since most op amps have a low frequency CMR of between 80 dB and 120 dB, it is clear that this circuit is only marginally useful for measuring CMRR (although it does an excellent job in measuring the matching of the resistors!).

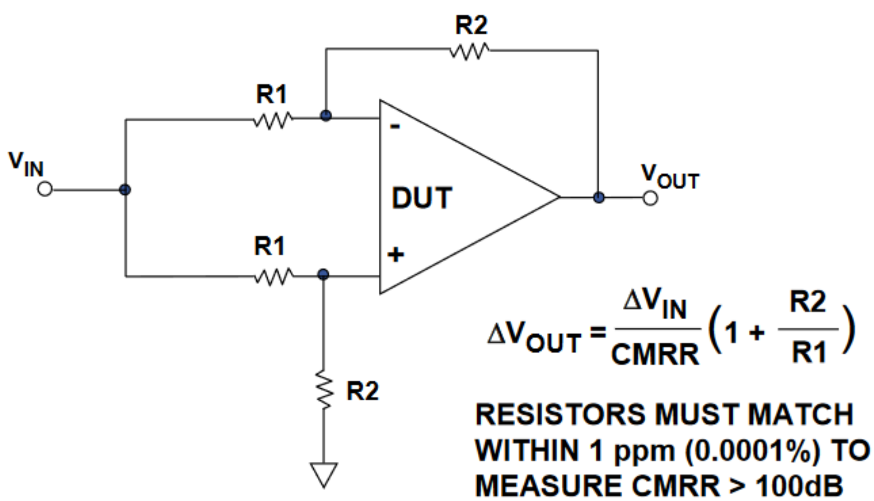



*Figure 3: Simple Common-Mode Rejection Ratio (CMRR) Test Circuit*

Courtesy: To solve this problem, I have used a reference which can be accessed by clicking the link: [0] [1] [2] [3] [4] [5]

M. Study table 5 (Pin Configuration and Function and Function Descriptions from the spec sheet for the Analog Devices Inc. AD8422 with this device operating at a nominal gain of 1000.

What value of gain resistor would you place across the $R_g$ pins to use this nominal gain? Select an appropriate resistor from the Digikey web site.

Now study figures 22 and 23 from the spec sheet for the AD8422, with this device operating at a nominal gain of 1000 at 10HZ.

Suppose your raw sensor signal is a sin wave with magnitude 4.6 mV and frequency 20,000 hz. and you are using the AD8422 amplifier to process it. The RF noise on the leads from the sensor to the data logger is 23 mV, what will be the percentage of noise on the amplified signal (i.e. the SNR)?

Sol. **19.819 Ω** and **0.002810 %**

To find the appropriate gain resistor value across the $R_g$ pins, I have used the below given data provided in the AD8422 spec sheet to reach a value:

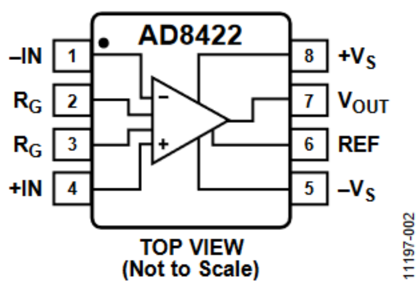



*Figure 3. Pin Configuration*

**Table 5. Pin Function Descriptions**

| Pin No. | Mnemonic | Description |
|---|---|---|
| 1 | –IN | Negative Input Terminal. |
| 2, 3 | $R_G$ | Gain Setting Terminals. Place resistor across the $R_G$ pins to set the gain. $G = 1 + (19.8\ k\Omega/R_G)$. |
| 4 | +IN | Positive Input Terminal. |
| 5 | $–V_S$ | Negative Power Supply Terminal. |
| 6 | REF | Reference Voltage Terminal. Drive this terminal with a low impedance voltage source to level shift the output. |
| 7 | $V_{OUT}$ | Output Terminal. |
| 8 | $+V_S$ | Positive Power Supply Terminal. |

Based on the data provided in the above table, the relation between the nominal gain and the gain resistor value is as under:
$G = 1 + (19.8\ k\Omega / R_g)$

Thus, solving this equation for G = 1000, we get,

$1000 = 1 + (19.8\ k\Omega / R_g)$

Thus, the value of the required gain resistor corresponding to a nominal gain value of 1000 is

Rg = 19.8 / (1000-1) = **19.819 Ω**

***The corresponding that I found on Digikey can be accessed by clicking here.***

Now, based on the figures 22 and 23 provided in the spec sheet corresponding to the operating frequency value of 20k Hz, the gain (dB) is approximately 55 dB and the CMRR (dB) value is approximately 105 dB.

Using the above obtained data, we calculate the absolute value of the differential gain:

$CMRR\ (dB) = 20 \times \log_{10} (A_{diff} / A_{comm})$ … [1]

Where, CMRR = Common Mode Rejection Ratio
$A_{diff}$ = Differential Mode Gain
$A_{comm}$ = Common Mode Gain

Based on the previously obtained data through figures 22 and 23,
$A_{diff}\ (dB) = 55\ dB$

Thus, $A_{diff}\ (dB) = 20 \log_{10} (A_{diff})$

$\therefore 55 = 20 \log_{10} (1 / A_{diff})$

$\therefore 55 / 20 = \log_{10} (1 / A_{diff})$

$\therefore 2.75 = \log_{10} (1 / A_{diff})$

$\therefore 10^{\wedge} (2.75) = A_{diff}$

$\therefore A_{diff} = \underline{562.341}$

CMRR = 105 dB

Thus, using Eq [1], we obtain, $105 = 20 \log_{10} (562.341 / A_{comm})$

$\therefore 105 / 20 = \log_{10} (562.341 / A_{comm})$

$\therefore 5.25 = \log_{10} (562.341 / A_{comm})$

$\therefore 10^{\wedge}5.25 = 562.341 / A_{comm}$

∴ $A_{comm}$ = 562.341 / (10^5.25) = 0.003162

Based on the value obtained for common mode gain, we can find the RF noise output for the instrumentation amplifier as below:

= $A_{comm}$ x (RF noise on the leads from the sensor to the data logger)

= 0.003162 x 23 mV = 0.0727 mV … [2]

Now, as for the raw sensor signal which is having a magnitude of 4.6 mV and a frequency of 20k Hz sine wave is applicable as a differential mode signal and therefore the output corresponding to this raw sensor signal will be as follows:

= differential signal x differential gain
= 4.6 mV x 562.341
= 2586.76 mV … [3]

Now, to calculate the percentage of noise, we divide equation [2] by equation [3] where equation [2] is the noise output and equation [3] is the amplified sensor signal.

Thus, percentage of noise on the amplified signal:

= Eq [2] / Eq [3]
= (0.0727/2586.76) x 100 %
= **0.002810 %**

Courtesy: To solve this problem, I have used the attached references: [1] [2]

N. How does Sensors Online magazine recommend that you perform digital noise measurements on a MEMS accelerometer, and minimize them accordingly?

Sol.

**Recommendation by Sensors Online Magazine regarding digital noise measurements**

While performing digital measurements, one should pay special attention to the Aliasing/Nyquist Theorem.

1. ***Thus, sensors to be tested should be bandwidth-limited accordingly.***

The required sampling frequency in accordance with the Nyquist Theorem is the Nyquist frequency ($f_N$) given by **Equation 3:**

$$f_N = 2 * f_{-3dB}$$

where:

$f_{-3dB}$ = low-pass filter cutoff frequency

2. ***To reconstruct the signal accurately, the Sensors Online magazine "recommends" that the sampling rate be between five to ten times the low-pass filter cut-off frequency.***

*Process for digital noise measurement:*

In an ideal ADC, the input analog voltage is increased and the ADC maintains a constant output code until a transition region is reached, at which point the ADC instantly jumps to the next code value and remains there until the next transition region is reached. In practice, an ADC has a certain amount of code transition noise and, therefore, a finite transition region width. Input-referred noise is generally characterized by examining a histogram of a number of output samples while the input to the ADC is held at a constant DC value.

3. ***To measure the input-referred noise, the input to the ADC needs to be heavily decoupled by setting the low-pass filter frequency to 50 Hz or lower and then a large number of samples can be collected and plotted as a histogram (around one million conversions are more than adequate for a low-noise ADC).***

Because the noise is approximately Gaussian, the standard deviation of the histogram is the RMS noise (**Figure 1**).

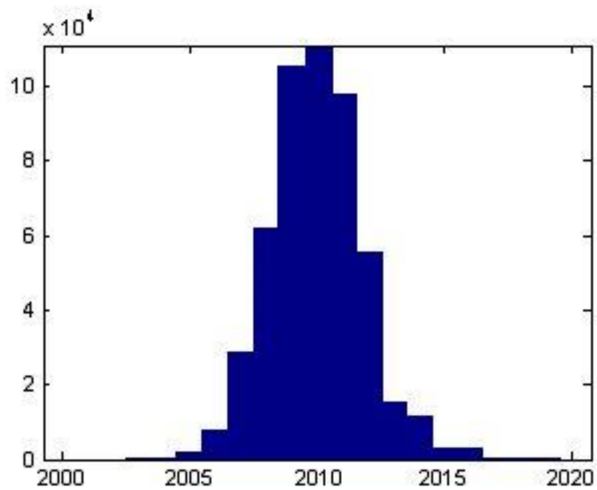



Figure 1. ADC output code

As we can see in Figure 1, there is also some inherent differential nonlinearity (DNL) associated with this ADC but, for the most part, it is still approximately Gaussian.

4. ***If there is significant DNL, but the results still follow a somewhat Gaussian distribution, then the standard deviation should be computed for several DC input voltage values and the results averaged.***

A code distribution that is significantly non-Gaussian could indicate a bad PC board layout, poor grounding techniques, or improper power supply decoupling.

**Reducing Noise**

1. ***The input-inferred noise could be reduced by placing a higher-order filter on the outputs or by reducing the bandwidth of the filter.***

Of course, one of the penalties for placing a higher-order filter on the outputs is an increase in the number of external components required on the printed circuit board. Likewise, one of the penalties for reducing the filter bandwidth is the increase in the startup/response time of the outputs. Depending on the application, this may result in the application having a sluggish response to motion, which can lead to a poor experience for the user.

2. ***Another technique that can be used to reduce noise is over-sampling and averaging.***

In most applications, digital data from the accelerometer are used in computations or in control functions. These data are obtained either directly from the accelerometer or after the analog data from the accelerometer have passed through an ADC. In either case, the system is acquiring accelerometer information at a particular sampling frequency. The optimal sampling rate will depend on the particular application. For example, a cell phone screen rotation application may have a sampling rate of 10 Hz while a hard-drive protection application may require a sampling rate of 1000 Hz.

3. ***Oversampling involves sampling a signal using a sampling frequency that is significantly higher than that of the Nyquist frequency.***

For example, the cell phone screen rotation application may be acquiring data at a 100 Hz sampling frequency. Every 10 samples are averaged together and that average value is reported at a 10 Hz frequency and used in the application to determine whether the screen has rotated. In this way, the noise in the acceleration signal is reduced.

4. ***If multiple samples (N) are taken of the same quantity with a random noise signal, then averaging those samples reduces the noise by a factor of 1/√N.***

Courtesy: To solve this problem, I have used the attached reference: [1]

O. What is an operational amplifier?

Sol.

An operational amplifier, or op amp, is a high-gain electronic voltage amplifier which is DC coupled and contains a differential input and, usually, a single-ended output. An op amp produces an output voltage which is often several times larger than the voltage difference between its input terminals. The differential inputs of an operational amplifier consist of a V− input and a V+ input. Op amps amplify the voltage difference in between the two, i.e. the differential input voltage.

An op amp is a voltage amplifying device. With the help of some external components, an op amp, which is an **active** circuit element, can perform mathematical operations such as addition, subtraction, multiplication, division, differentiation and integration. If we look at a general op amp package (innards to come in a later tutorial) such as the ubiquitous 741, we'll notice a standard 8-pin DIP (dual in-line package):

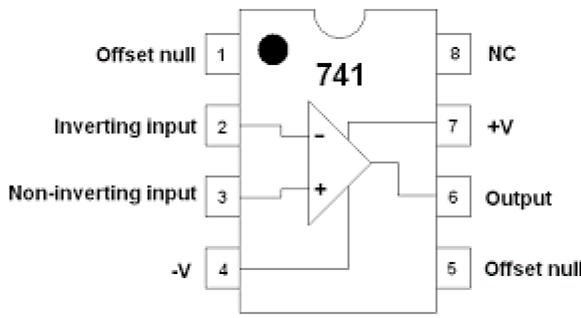
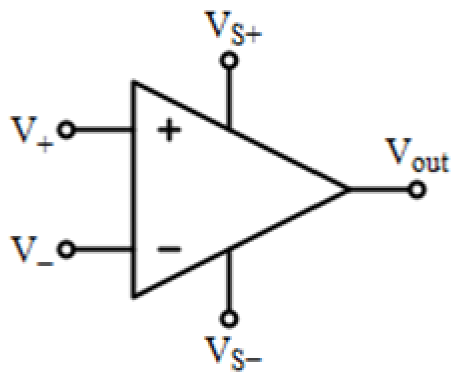



*Photo courtesy of Learning About Electronics*

We are mainly concerned with five of the pins. The circuit symbol for an op amp is a triangle with five pins shown below.




- $V_+$ : non-inverting input
- $V_-$ : inverting input
- $V_{out}$ : output
- $V_{S+}$ : positive power supply
- $V_{S-}$ : negative power supply

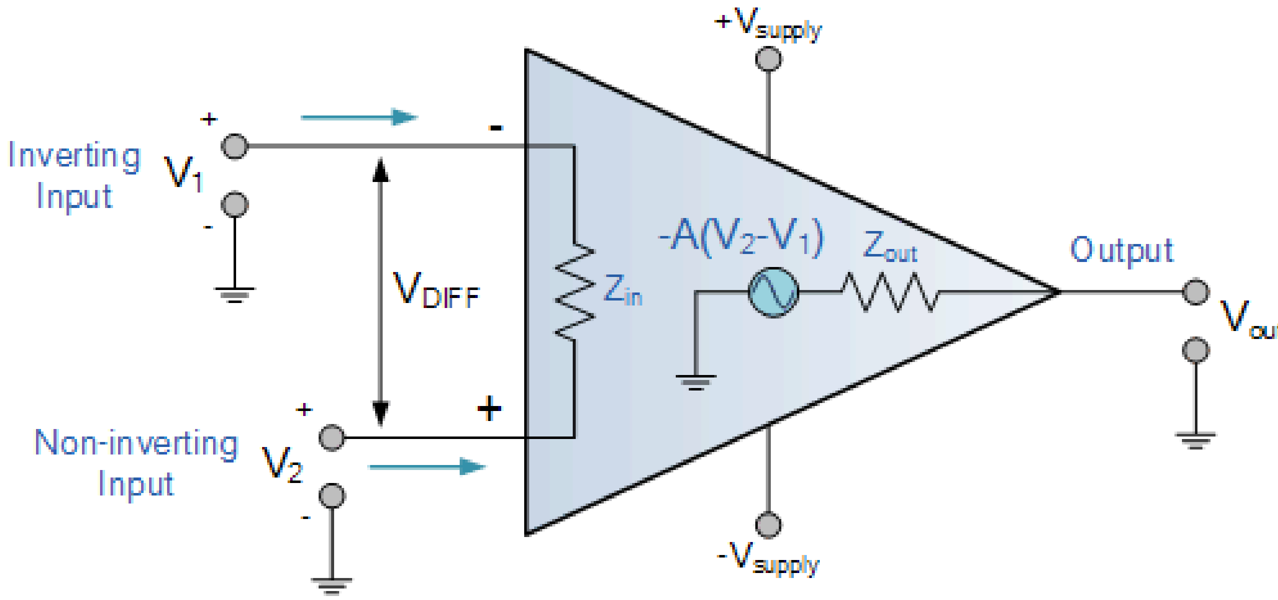
*Photo courtesy of Virtual Labs*

An op amp has a wide range of uses and, depending how each pin is connected, the resulting circuit can be some of the following (this is by no means a comprehensive list):

- Comparator
- An Inverting Amplifier such as a summing amplifier
- A Non-Inverting Amplifier such as a voltage follower
- Difference Amplifier
- Differentiator or Integrator
- Filter
- Peak Detector
- Analog-to-Digital Converter
- Oscillator

An operational amplifier is a very close approximation to a perfect amplifier with an infinite gain. In reality op-amps do not quite attain perfection, but with gains often in the region of 100 000 or more, they are sufficiently close.

The operational amplifier has two inputs. One is called the inverting input and is marked with a "-" sign on circuit schematic diagrams. The other is the non-inverting input and this is marked with a "+" sign.




Operational amplifier circuit symbol

The two inputs gain their names from the way in which they amplify the signals:

- ***Non-inverting input:*** The operational amplifier non-inverting input is marked by a "+" sign on the circuit diagram. It is found that a positive voltage applied to the non-inverting input will produce a positive swing at the output.
- ***Inverting input:*** The operational amplifier inverting input is marked by a "-" sign on the circuit diagram. A positive voltage applied to the inverting input will produce a negative swing at the output.

If the same voltage is applied to both inputs together then there should be no change at the output. In fact the output is proportional to the difference between the inverting and non-inverting inputs. It is for this reason that these amplifiers are often called differential amplifiers.

Like any electronics circuit, those using operational amplifiers need to have a power supply. Normally op-amps are supplied using dual, i.e. positive and negative supplies. Additionally the supply lines are often not shown as they add confusion to the circuit diagram.

In most cases the operational amplifier will only need five connections for its operation - inverting, non-inverting, output and the two power rails. Very occasionally a further three may be used. These are usually for the "offset null" capability. This is used to reduce any DC offsets that may be present, and for most applications these can be ignored and left disconnected.

Courtesy: To solve this problem, I have used a reference which can be accessed by clicking the link: [0] [1] [2] [3] [4] [5] [6]*