

Schmartboard OPAMP Board Applications

Part # 710-0011-01

The OPAMP Board makes it very easy to assemble, and experiment with, op amp circuits.

It uses an OPA172 operational amplifier whose main features make it very suitable for experimenting.

These features are,

- Wide Supply Range: +4.5 V to +36 V, ± 2.25 V to ± 18 V
- Low Offset Voltage: ± 0.2 mV
- Gain Bandwidth: 10 MHz
- Input Range Includes the Negative Supply
- Input Range Operates to Positive Supply
- Rail-to-Rail Output
- Low Noise: 7 nV/ $\sqrt{\text{Hz}}$
- High Common-Mode Rejection: 120 dB
- Low Input Bias Current: ± 8 pA
- Low Quiescent Current: 1.6mA per Amplifier

The OPAMP Board can be operated with single or dual power supplies and the 10MHz bandwidth makes is sufficient for most experimenter's circuits.

The applications in this app note all use ± 12 V power supplies.

The OPA172 data sheet can be downloaded from the SchmartBOARD website.

A note about the OPA172 is that for low closed loop gains there can be a little overshoot when a fast rising or falling edge pulse is applied at the input. This overshoot can be eliminated by putting a small capacitor, about 22pF, across the feedback resistor. This can be accomplished by putting this capacitor in the C1 location and a zero ohm resistor, or a piece of wire, in the place of R7.

For an example of the overshoot, run the InvertingOPA172noCap.asc or the

Non-InvertingOPA172noCap.asc LTspiceIV simulation. This will show the overshoot using 10ns rise and fall time pulse input with no feedback capacitor. Once the simulation is finished, click the cursor on the output pin of the op-amp. An output pulse with overshoot will be shown on the oscilloscope window.

Some simulation files have two versions, one with the 22pF feedback cap and one without the cap. To see the effect of having a 22pF feedback capacitor, run both files using LTspiceIV and look at the output of the OpAmp. Notice there is no longer overshoot in the output with the cap. Also try using the "noCap" files and changing the rise time and fall time of the inputs to see how they affect the overshoot.

This capacitor can be added for all the applications described here if necessary. For lower rising and falling input signals this won't be necessary. In general, if this capacitor and a zero ohm resistor are installed, it will not negatively affect any of the applications described here.

In the examples, R_p is used to balance the voltage drop across the resistors connected to the positive input of the op-amp with the voltage drop across the resistors connected to the negative input to the op-amp. The OPA172 input bias currents are relatively low so any imbalance between the voltage drops at the negative and positive inputs are minimal. A zero ohm resistor, or a wire, could be instead of a resistor without much output offset error due to this imbalance.

Op-Amp Resources

Following are some resources for getting more information on applying operational amplifiers.

<http://www.ti.com/ww/en/bobpease/assets/AN-31.pdf>

<http://www.ti.com/lit/an/sboa092a/sboa092a.pdf>

<http://my.ece.ucsb.edu/York/Bobsclass/2C/Tutorials/App%20notes/an-4.pdf>

http://www.analog.com/library/analogDialogue/archives/39-05/Web_Ch1_final_R.pdf

Circuit Simulation using LTspiceIV

These examples have SPICE simulation files on the SchmartBOARD website. These use LTspiceIV which can be downloaded for free from the Linear Technology website. The following describe the setup necessary to run these files.

There are two types of simulation used in this app note.

1) Transient analysis.

This calculates the circuit response to various transient waveforms such as a pulse, pulses, square waves, triangle waves etc. The output is similar to what would be seen when observing the various points in the circuit when using an oscilloscope.

2) Frequency analysis.

This calculates the general frequency response of the circuit. The output is a BODE plot as would be seen when using a spectrum analyzer. It shows the gain and phase response to a swept frequency sine wave input.

Due to a limitation of the OPA172.LIB circuit description, two different models used for these simulations.

1) OPA172.LIB

The OPA172.LIB model is used for transient simulations. When using this model, the transient simulation results fit actual real world transient results of the OPA172. Its limitation is that it does not work correctly for AC analysis. It will seem to do AC simulations but the results are not correct. The only thing found to be missing are two parallel diodes across the plus and minus inputs of the OPA172. These will not affect the simulation results unless the OPA172 is used in a circuit where it is used as a comparator rather than a linear amplifier. These diodes are shown in the data sheet but for some reason were not included in the model. 1N914 diodes can be used. The simulations

results with these diodes will agree much more with actual bench measurements when using the OPA172 as a comparator.

2) LTspiceIV's internal UniversalOpamp2.asy.

After modifying some parameters of this model, the AC simulations will be very close to what would be measured using an actual spectrum analyzer.

It can be used for transient simulations but it will not show non-ideal real world responses of the OPA172 such as overshoot when a fast rise and/or fall time transient signal is applied.

Installing and Running LTspiceIV

First download the LTspiceIV from <http://www.linear.com/solutions/ltspice>

Install LTspiceIV by double clicking on the downloaded .exe file

Once it is installed, it will ask if you want to update it. Answer Yes.

Download OPA172.LIB from the SchmartBOARD website.

Put OPA172.LIB file in the C:\Program Files (x86)\LTC\LTspiceIV\lib\sub directory. Transient analysis simulations will not run without doing this.

The simulation files included with this app note can then be run either by opening the .asc simulation file or by opening it from within LTspice.

Once the simulation file is opened, select **Run** in the **Simulate** menu.

After the simulation is finished, an oscilloscope looking screen will pop up if a transient analysis is run or a spectrum analyzer type window will open if it is an AC analysis is run. In either case, click the cursor on the point in the circuit that you want to see. That waveform results at that point in the circuit will show up in the appropriate window.

There will be one or two type of simulation files for each application. For transient simulation, the last four characters in the filename will be TRAN and for AC simulation the last two characters in the filename will be AC.

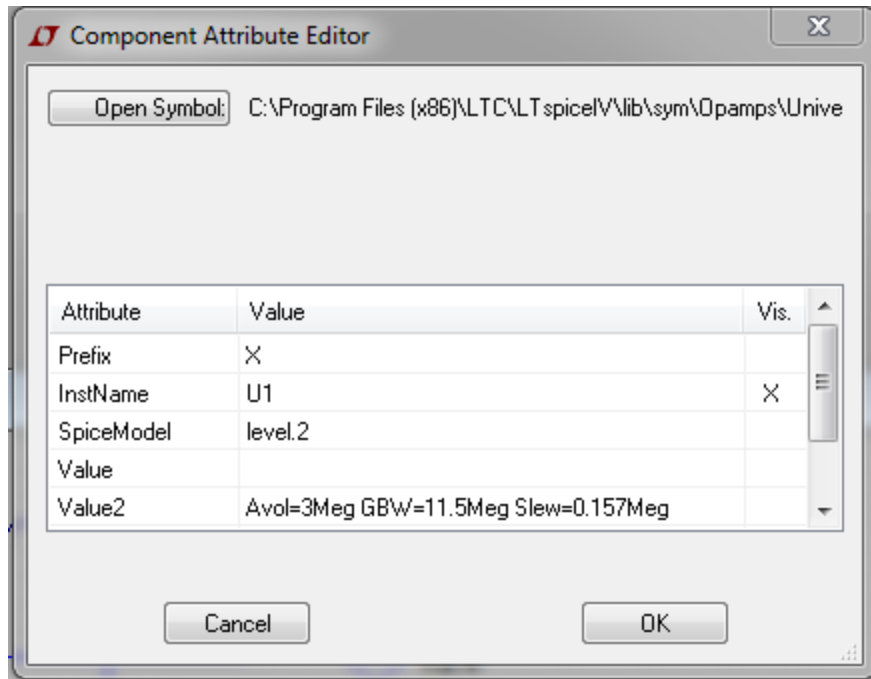
For example,

InvertingOPA172noCapTRAN.asc is a simulation file for a transient simulation,

InvertingOPA172noCapAC.asc is a simulation file for AC analysis.

The appropriate model is already included in the files for this app note. If you are generating your own AC analysis circuits using the OPA172 the UniversalOpamp2.asy parameters must be changed. These parameters have been changed for the files used here.

The parameters that should be changed when generating your own AC simulation circuits are,



The line,

Avol=1Meg GBW=10Meg Slew=10Meg should be changed to,

Avol=3Meg GBW=11.5Meg Slew=0.157Meg.

These can be accessed by double clicking on the on the op amp symbol.

Inverting Amplifier

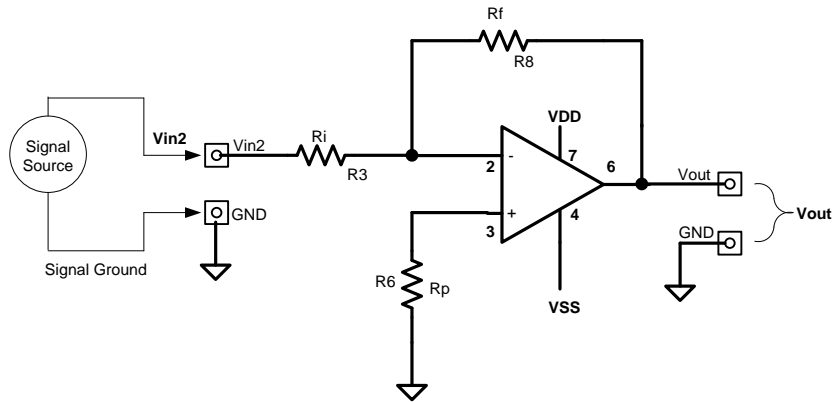


Figure 1, Inverting Amplifier.

$$V_{out} = -V_{in} \times \frac{R_f}{R_i}$$

$$Gain = -\frac{V_{out}}{V_{in}} = -\frac{R_f}{R_i}$$

$R_p = R_f \parallel R_i$, the parallel combination of R_f and R_i ,

$$R_p = \frac{R_f \times R_i}{R_f + R_i}$$

An Implementation of Inverting Amplifier Using the OpAmp Board

SPICE simulation files;

InvertingOPA172noCapTRAN.asc
 InvertingOPA172noCapAC.asc
 InvertingOPA172withCapTRAN.asc
 InvertingOPA172withCapAC.asc

If there are problems running TRAN simulations, verify the OPA172.LIB file is in the C:\Program Files (x86)\LTC\LTspiceIV\lib\sub directory.

Design

Using R_3 for R_i , R_8 for R_f , and R_6 for R_p ,

$$V_{out} = -V_{in2} \times \frac{R_8}{R_3}$$

$$Gain = -\frac{V_{out}}{V_{in2}} = -\frac{R_8}{R_3}$$

$$R_6 = \frac{R_8 \times R_3}{R_8 + R_3}$$

Example with Gain = -1,

Let $R8 = 10K$,

$$R3 = -Gain \times R8 = -(-1) \times 10K = 10K$$

$$R6 = \frac{R8 \times R3}{R8 + R3} = 5K$$

The standard value of 5.1K will be close enough.

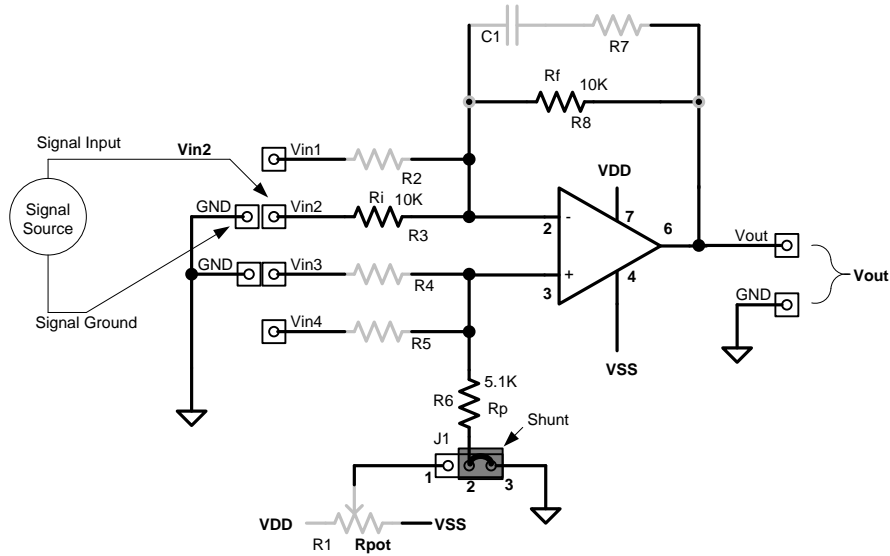


Figure 2, Implementation of Inverting Amplifier.

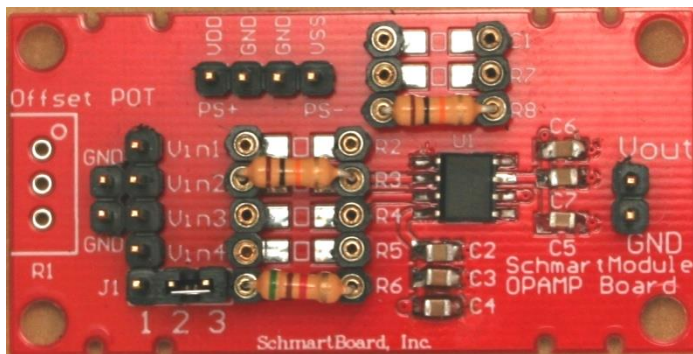


Photo 1, Implementation of Inverting Amplifier

Non-Inverting Amplifier

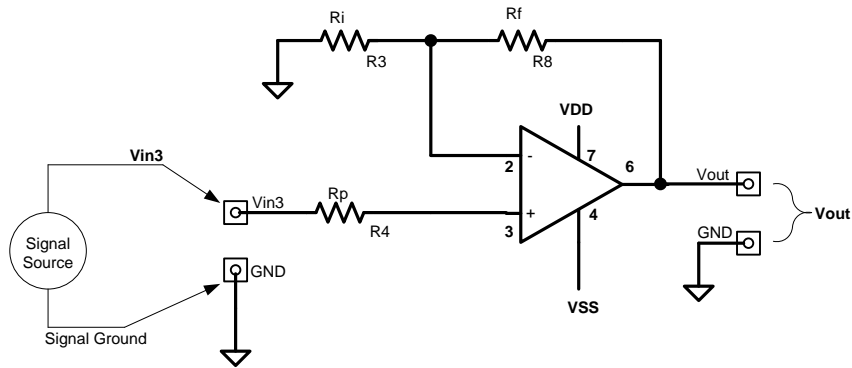


Figure3, Non-Inverting Amplifier.

$$V_{out} = V_{in3} \times \left(1 + \frac{R_f}{R_i}\right)$$

$$Gain = \frac{V_{out}}{V_{in3}} = 1 + \frac{R_f}{R_i}$$

$R_p = R_f \parallel R_i$, the parallel combination of R_f and R_i ,

$$R_p = \frac{R_f \times R_i}{R_f + R_i}$$

An Implementation of a Non-Inverting Amplifier Using the OpAmp Board

SPICE simulation files;

Non_InvertingOPA172noCapTRAN.asc
Non_InvertingOPA172noCapAC.asc
Non_InvertingOPA172withCapTRAN.asc
Non_InvertingOPA172withCapAC.asc

If there are problems running TRAN simulations, verify the OPA172.LIB file is in the C:\Program Files (x86)\LTC\LTspiceIV\lib\sub directory.

Design

Using R3 for R_i , R8 for R_f , and R4 for R_p .

$$V_{out} = V_{in3} \times \left(1 + \frac{R8}{R3}\right)$$

$$Gain = \frac{V_{out}}{V_{in3}} = 1 + \frac{R8}{R3}$$

$R_p = R_f \parallel R_i$, the parallel combination of R_f and R_i ,

$$R_p = \frac{R_f \times R_i}{R_f + R_i}$$

Let Gain = 2, $R_8 = 10K$,

$$R_3 = \frac{R_8}{(Gain - 1)} = \frac{R_8}{(2 - 1)} = R_8 = 10K$$

$$R_4 = \frac{R_8 \times R_3}{R_8 + R_3} = 5K$$

The standard value of 5.1K will be close enough for R_4 .

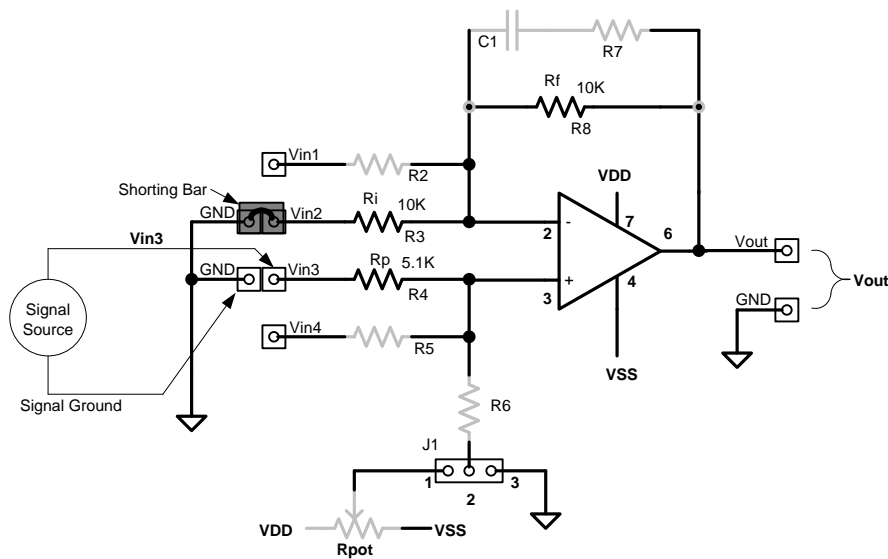


Figure 4, Implementation of Non-Inverting Amplifier.

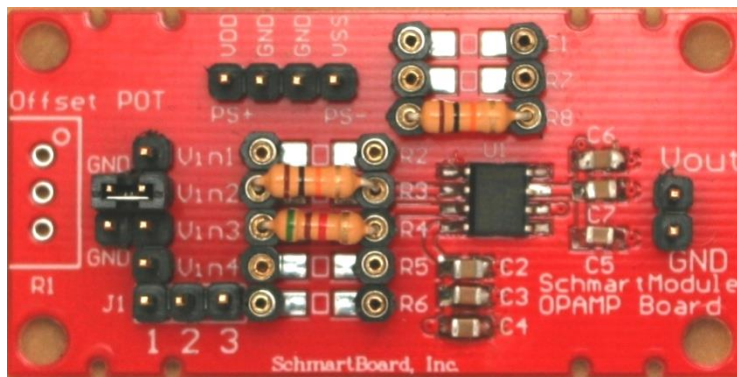


Photo 2, Implementation of Non-Inverting Amplifier

Differential Amplifier

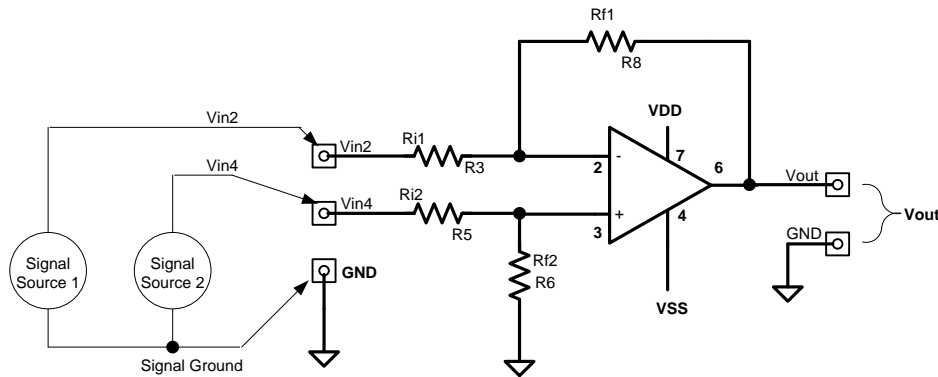


Figure 5, Schematic of Differential Amplifier.

$$V_{out} = \left[\frac{(R_{i1} + R_{f1})}{(R_{i2} + R_{f2})} \right] \times \frac{R_{f2}}{R_{i1}} \times V_{in4} - \frac{R_{f1}}{R_{i1}} \times V_{in2}$$

For $R_{f1}=R_{f2} = R_f$ and $R_{i1}=R_{i2}=R_i$,

$$V_{out} = (V_{in4} - V_{in2}) \times \frac{R_f}{R_i}$$

$$Gain = \frac{R_f}{R_i}$$

An Implementation of Differential Amplifier Using the OpAmp Board

SPICE simulation files;

DifferentialOPA172noCapTRAN.asc

DifferentialOPA172withCapTRAN.asc

If there are problems running TRAN simulations, verify the OPA172.LIB file is in the C:\Program Files (x86)\LTC\LTspiceIV\lib\sub directory.

Design

Let $R_8 = R_6 = R_f = 10K$, $R_2 = R_5 = R_i = 5.1K$,

$$Gain = \frac{R_f}{R_i} = \frac{10K}{5.1k} = 1.96 \cong 2$$

Let Gain = 2,

$$V_{out} = (V_{in4} - V_{in2}) \times 2$$

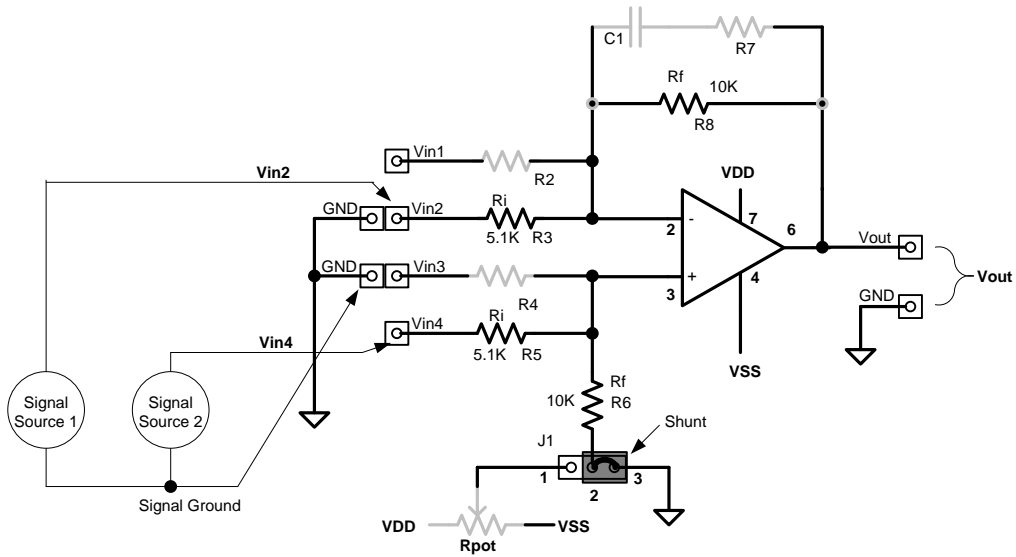


Figure 6, Implementation of a Differential Amplifier.

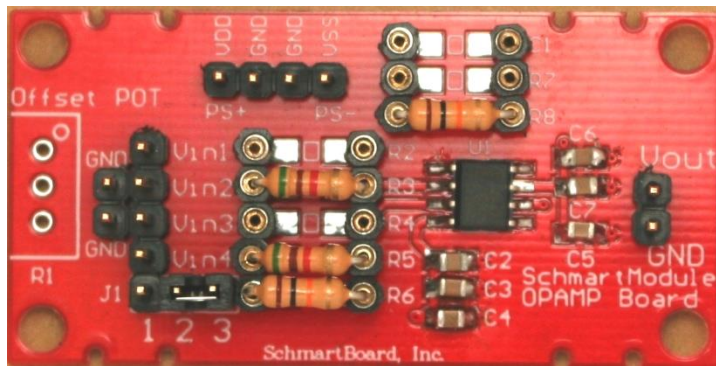


Photo 3, Implementation of a Differential Amplifier

Unity Gain High Input Impedance Buffer

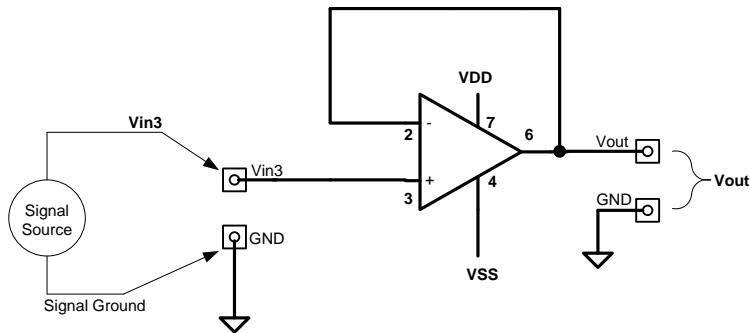


Figure 7, Schematic of Unity Gain High Input Impedance Buffer.

$$V_{out} = V_{in3}$$

Since the input bias current between the positive and negative inputs are low, no feedback resistor nor input resistor is shown.

An Implementation of Unity Gain High Input Impedance Buffer Using the OpAmp Board

SPICE simulation files;

UnityGainBufferOPA172TRAN.asc

UnityGainBufferOPA172AC.asc

If there are problems running TRAN simulations, verify the OPA172.LIB file is in the C:\Program Files (x86)\LTC\LTspiceIV\lib\sub directory.

R8 = R4 = zero ohm resistor or wire.

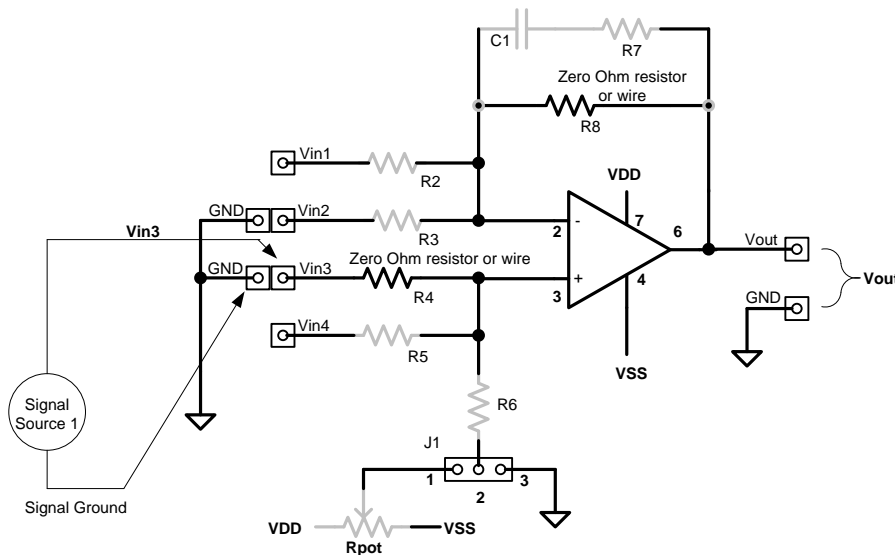


Figure 8, Implementation of Unity Gain High Input Impedance Buffer.

Inverting Summing Amplifier

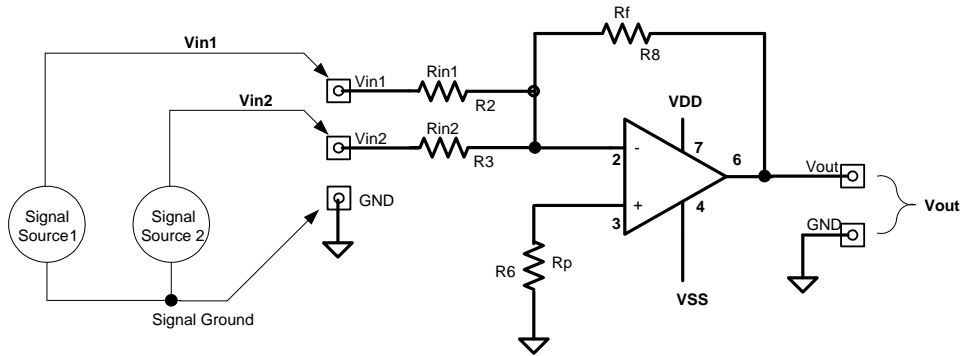


Figure 9, Schematic of Inverting Summing Amplifier.

$$V_{out} = -R_f \times \left(\frac{V_{in1}}{R_2} + \frac{V_{in2}}{R_3} \right)$$

$R_p = R_f || R_{in1} || R_{in2}$, the parallel combination of R_f , R_{in1} and R_{in2}

$$R_p = \frac{1}{\left(\frac{1}{R_f} + \frac{1}{R_{in1}} + \frac{1}{R_{in2}} \right)}$$

An Implementation of Inverting Summing Amplifier Using the OpAmp Board

SPICE simulation files;

SummingInvertingOPA172noCapTRAN.asc

SummingInvertingOPA172withCapTRAN.asc

If there are problems running TRAN simulations, verify the OPA172.LIB file is in the C:\Program Files (x86)\LTC\LTspiceIV\lib\sub directory.

Design

Let $R_f=R_8=5.1K$, $R_{in2}=R_2=10K$, $R_{in3}=R_3=10K$, $R_p=R_6$,

$$V_{out} = -5.1K \times \left(\frac{V_{in1}}{10K} + \frac{V_{in2}}{10K} \right)$$

$$R_6 = \frac{1}{\left(\frac{1}{5.1K} + \frac{1}{10K} + \frac{1}{10K} \right)} \cong 2.5K$$

Since the nearest standard 5% value to 2.5K is 2.7K,

$R_6 = 2.7K$, a 5.1K could be used with little effect offset effect.

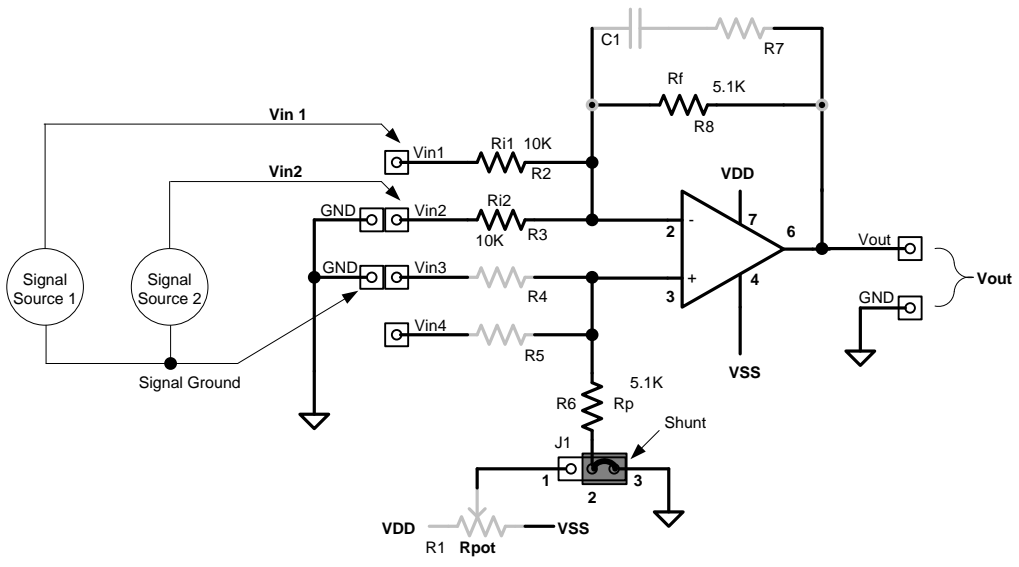


Figure 10, Implementation of Inverting Summing Amplifier.

Non-Inverting Summing Amplifier

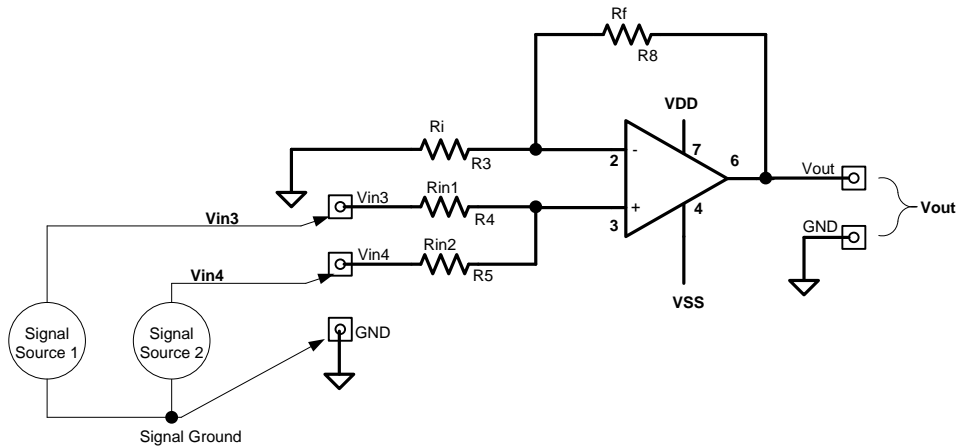


Figure 11, Schematic of Non-Inverting Summing Amplifier.

General equation for Non-Inverting Summing Amplifier,

$$V_{out} = \frac{V_{in3} \times R_{in2} + V_{in4} \times R_{in1}}{R_{in1} + R_{in2}} \times \left(1 + \frac{R_f}{R_i}\right)$$

This simplifies greatly when $R_{in1}=R_{in2}$,

$$V_{out} = \frac{V_{in3} + V_{in4}}{2} \times \left(1 + \frac{R_8}{R_3}\right)$$

$$V_{out} = \frac{[(V_{in3} \times R_5 + V_4 \times R_4) \times \left(1 + \frac{R_8}{R_3}\right)]}{R_4 + R_5}$$

An Implementation of of Non-Inverting Summing Amplifier Using the OpAmp Board

SPICE simulation files;

SummingNonInvertingOPA172noCapTRAN.asc

SummingNonInvertingOPA172withCapTRAN.asc

If there are problems running TRAN simulations, verify the OPA172.LIB file is in the C:\Program Files (x86)\LTC\LTspiceIV\lib\sub directory.

Design

Let $R_4=R_{in1}$, $R_5=R_{in2}$ and $R_4=R_5$, $R_8=10K$, $R_3=10K$,

$$V_{out} = \frac{V_{in3} + V_{in4}}{2} \times \left(1 + \frac{R_8}{R_3}\right) = \frac{V_{in3} + V_{in4}}{2} \times \left(1 + \frac{10K}{10K}\right)$$

$$V_{out} = \frac{V_{in3} + V_{in4}}{2} \times (1 + 1) = \frac{V_{in3} + V_{in4}}{2} \times (2)$$

$$V_{out} = V_{in3} + V_{in4}$$

Since R4 and R5 drop out of the equation, they can be anything. To keep the offset error to a minimum due to input bias current, $R5 || R4$ should be close to $R8 || R3$. Using $R4=5.1K$, $R5=5.1K$ is within a factor of two, close enough for this op-amp.

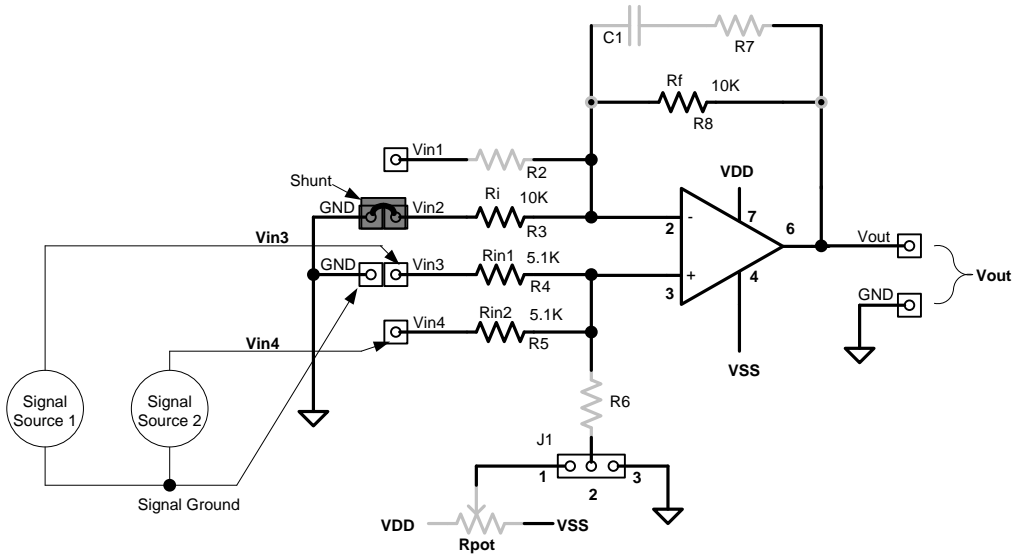


Figure 12, Implementation of Non-Inverting Summing Amplifier.



Schmartboard, Inc.
 37423 Fremont Boulevard
 Fremont, CA 94536
 510-744-9900 (Phone)
 510-744-9909(Fax)

info@schmartboard.com
 www.schmartboard.com
 www.facebook.com/schmartboard
 www.twitter.com/schmartboard