# Low-Cost Alternative to a Function Generator: The Bubba Oscillator

by Phillip Johnston
4 March 2012

## Introduction

When testing the functionality of a circuit or system, it is often necessary to utilize an input waveform with specific frequency and voltage characteristics in order to characterize the response. In lab environments, a function generator is used for this type of testing, and is able to generate various waveforms at a range of input voltages and input frequencies.

However, if you are a hobbyist or student, access to a function generator may not be possible or convenient. Additionally, the cost of most function generators can be prohibitive to those operating on a budget. However, simple, low-cost, quick-to-assemble oscillator circuits can be used instead of function generators for most tests, and can be assembled with components commonly found on hand, such as op amps, resistors, and capacitors.

One such oscillator is the Bubba oscillator. By using the Bubba oscillator, a sine and cosine wave can be generated at a specified frequency by simply using four op amp RC circuits in series. This circuit also has the benefit of providing low harmonic distortion, reducing the effect of harmonic presence on the signal.

Using these effects, a sine wave signal can be feasibly generated at any desired frequency by selecting the appropriate components.

## The "Bubba" Oscillator

The "Bubba" oscillator is a phase-shift oscillator that utilizes four op amp RC networks in series. Each op amp provides 45° phase shift, for a total of 180° phase shift through the entire circuit. The circuit is able to provide a stable sine wave at the specified frequency, and, if desired, the corresponding cosine wave.

While the ability to generate a sine waveform at any frequency is an appealing feature of the Bubba oscillator, the primary advantage of using it over other oscillator circuits is its low total harmonic distortion (THD). In general, THD is a measure of how much content a circuit adds at the harmonics of the fundamental frequency. This is especially important in audio circuits, where lower harmonic distortion allows for a more accurate reproduction of the signal, though any application requiring a high fidelity signal should seek to minimize THD.

Finally, while crystals and resonators are cheap, easy to acquire, and provide more precision, setting up a circuit to utilize these devices is often much more complicated than simply assembling an oscillator circuit, especially when methods to divide a clock to the desired frequency are considered. The Bubba oscillator provides the greatest time-savings for arbitrary sine wave generation needs without using a function generator.

Table 1 shows a list and quantity of parts necessary to design a Bubba oscillator circuit.

**Table 1.** Parts required for Bubba Oscillator Construction

| Part | Quantity |
|---|---|
| 1.4M Resistor | 1 |
| 350K Resistor | 1 |
| 10K Resistors (Voltage divider) | 2 |
| Calculated Resistance | 4 |
| Selected Capacitance | 4 |
| Op Amp | 4 |

First, the desired frequency for the oscillator must be decided upon. Using this frequency, it is common practice to fix the capacitor value, leaving only one variable to work with. Typically, values of 1nF, 0.01µF, or 0.1µF are selected for this purpose. Once the capacitance has been selected, the resistor value can be calculated using Equation 1 below.

**Equation 1.** Formulae relating the oscillator frequency with resistor and capacitor values.

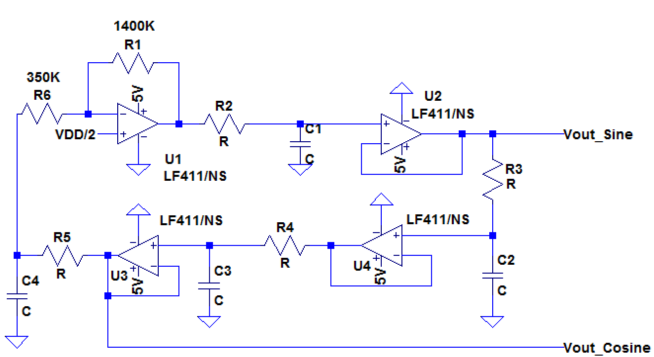
$$\omega_0 = \frac{1}{RC} \rightarrow R = \frac{1}{2\pi f_0 C}$$

As a caveat, be aware that each op amp has a particular frequency response. It is up to you to ensure that the frequency you have selected is compatible with the op amp in use. Frequency response information can commonly be found in the op amp's datasheet.

The Bubba oscillator is able to run on both unipolar (e.g., 5V) and bipolar (e.g., +15V, -15V) supplies. While higher voltage and a slightly better THD will typically be seen with a bipolar power supply, a unipolar design is detailed in this application note, with the thought that hobbyists and those on a budget may not have the capability of providing bipolar voltages. However, if a bipolar implementation is desired, simply replace 5V with the desired positive supply voltage and connect the negative power supply terminals of the op amp to the negative supply voltage.

Once the components have been selected and the power design topology decided upon, the circuit should be assembled as shown in Figure 1. The 'R' values used in the circuit represent the calculated resistance, and the 'C' values are the selected capacitance.

**Figure 1.** Schematic of the Bubba Oscillator. R corresponse to the calculated resistance, and C corresponds to the selected capacitance.




The resistors for the first inverting op-amp are selected to have a gain of four. Getting as close to this value as possible is desirable; gain slightly above four will cause the circuit to oscillate at a different frequency than desired, and gain less than four will not be high enough to sustain oscillation. While 1.4MΩ and 350KΩ were chosen, other values may be substituted. However, ensure that the resistor values used are large, as the noise generated by these components is used to start the oscillation.

Since each of the op amps shifts the output by 45°, it is possible to capture both a sine wave and a cosine wave from this oscillator circuit. By selecting the outputs at the nodes shown above in Figure 1, two stable outputs are found with 90° phase shift between them. Additionally, there will be a small voltage difference between the two waveforms (in the range of a few hundred millivolts).

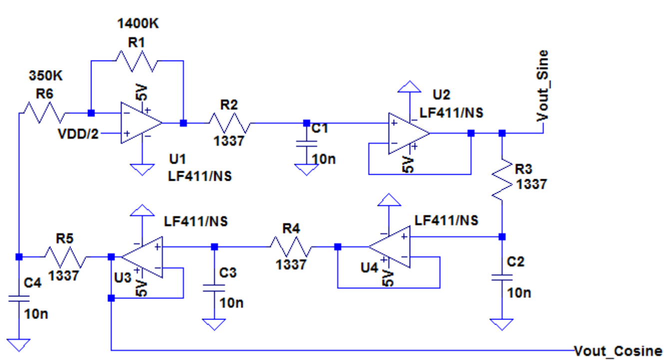
Furthermore, any two nodes with two op amps in between them may be utilized to provide sine and cosine waves. The selected nodes above are chosen for signal integrity.

## Example Implementation

In order to test the low pass filter of an in-house design, a Bubba oscillator was assembled to provide an input test signal. As our design under test is characterized by a 10 kHz cutoff frequency, an 11.9 kHz was selected in order to place it well into the stop band of the circuit.

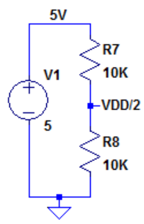
By fixing the capacitor value at 10 nF, the resistor value is calculated to be 1337 ohms. These components yield an expected frequency of 11.904 kHz. Using these calculated values, a circuit was assembled in LTSpice to verify the design via simulation, shown in Figure 2.

**Figure 2.** Bubba oscillator circuit used to generate an 11.9kHz sine wave output.




The 5V and VDD/2 nets were attached to the circuit as shown below in Figure 3.

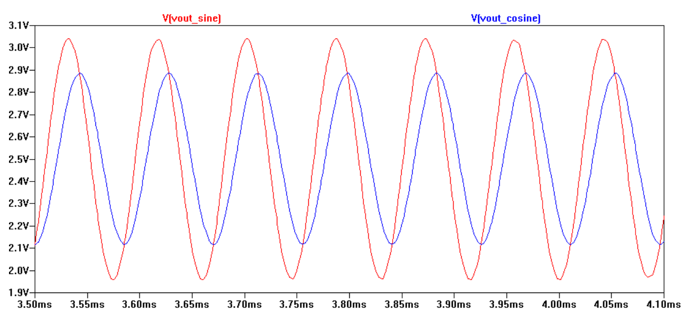
**Figure 3.** Circuit showing the attachment of 5V and VDD/2. VDD/2 is attained by using a voltage divider.




Using a voltage divider with equal valued resistors will provide the VDD/2 input to the circuit. Choose components in the 1KΩ-10KΩ range to reduce current flow through the voltage divider.

Using transient analysis, shown in Figure 4 below, the oscillator was shown to output the waveforms with the desired 90° phase shift.

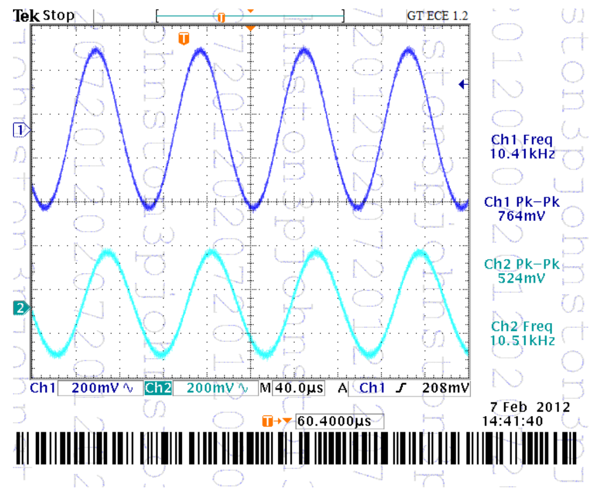
**Figure 4**. Plot showing the sine and cosine outputs of the Bubba oscillator. The two waveforms are 90 degrees out of phase, as is expected by picking outputs with two op amps in between.




Two things should be noticed about the above plot: both waveforms have a DC offset of 2.5V, which is VDD/2. Also, the peak-to-peak value of the cosine wave is smaller than the peak-to-peak value of the sine wave. This is expected, and both the DC offset and peak-to-peak voltages can be altered by using additional filtering on the output lines.

After simulating the circuit and verifying that the design meets our specifications, a prototype was built on a breadboard to analyze the THD and behavior of the circuit under real world conditions.

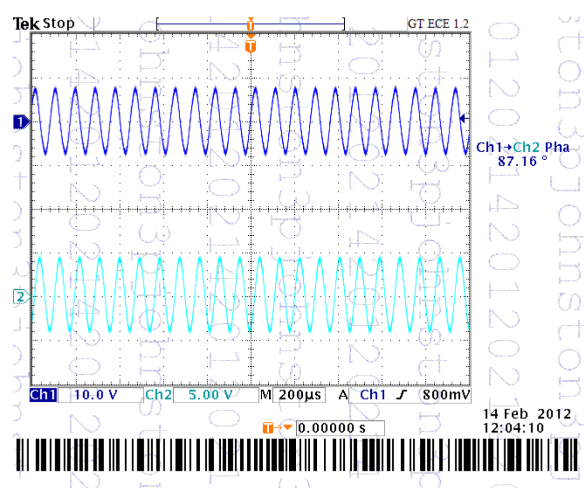
**Figure 5.** Oscilloscope capture showing the sine (Ch. 1) and cosine (Ch 2.) output of the Bubba Oscillator circuit.




The frequency output of the prototype circuit was measured to be about 10.45kHz. This is much lower than the desired 11.9kHz, due to the rounding of resistances and the use of 20% tolerance capacitors and 5% tolerance resistors. A more accurate frequency may be achieved with higher precision components, through the use of multiple resistors in series, or through the addition of potentiometers in series, which can be used to match the specified values more closely.

After verifying the frequency of our design, the phase shift between the two waveforms was measured to ensure they meet the design specification, shown in Figure 6.

**Figure 6.** Oscilloscope capture showing the phase shift between the sine and cosine outputs. Note that the phase shift is close to 90°.

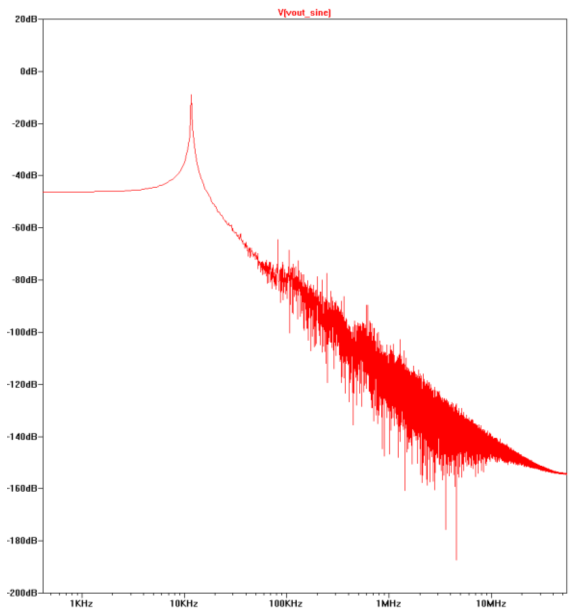



While not exactly 90°, the measured 87.16° phase shift produces only 3.16% error, which is quite acceptable under real world conditions, and close enough to not be noticeable in many situations.

## Total Harmonic Distortion

In order to analyze the Total Harmonic Distortion (THD) of the circuit, a Fast Fourier Transform (FFT) must be performed on each signal. Figure 7 shows the frequency spectrum of the simulation waveform after performing an FFT.

**Figure 7.** Fast Fourier Transform for the oscillator circuit. Note the single peak around 11.9kHz. Additionally, note that there are no peaks beyond the fundamental frequency.




Once the FFT is performed, the THD can be calculated using Equation 2.

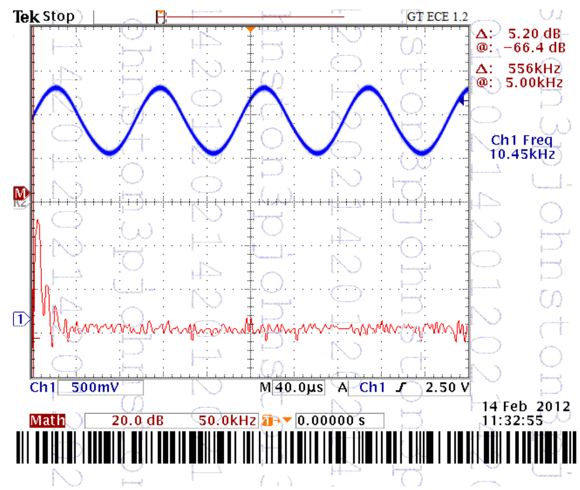
**Equation 2.** Method of calculating THD using RMS voltages

$$THD = \frac{(V_2^2 + V_3^2 + \cdots + V_n^2)^{\frac{1}{2}}}{V_1}$$

As the simulation uses ideal op amps, and as there is only one peak at 11.9kHz, the THD for the simulated circuit is zero. However, in the real world, there will be a small amount of distortion present, as the components used are not ideal.

Figure 8 shows the FFT of the sine waveform, as measured on the oscilloscope. Note that there are two harmonics present in addition to the fundamental frequency in this waveform.

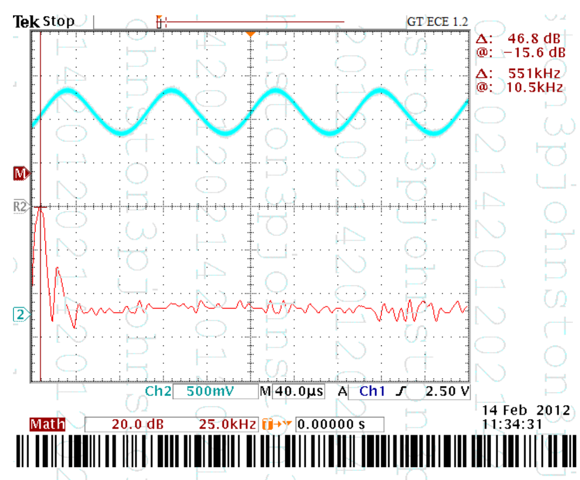
**Figure 8.** Fast Fourier Transform of the sine output. Note three distinct peaks on the FFT waveform; these correspond to the fundamental frequency and the first and second harmonics.




Using voltage measurements made for each of the harmonics present, the THD of the sine output was measured to be 3.7% (4.07% when a bipolar supply was used).

Figure 9 shows the FFT of the cosine waveform, as measured on the oscilloscope.

**Figure 9.** Fast Fourier Transform of the cosine output. Note two distinct peaks on the FFT waveform.




Using voltage measurements made for each of the harmonics present on the cosine FFT, the THD of the cosine output was calculated to be 2.57% (1.9% using a bipolar supply).

Overall, each output will provide you a sine or cosine waveform with very little distortion 9less than 5% THD).

## Additional Uses

The Bubba oscillator can be applied in many ways, other than as a simple replacement for a function generator.

The Bubba oscillator is most useful in prototyping, where arbitrary frequencies can be generated at will, allowing a wide range of applications and designs to be tested and hammered out before more low-power or more accurate alternatives are selected to provide the final clocking scheme.

Examples of such additional uses are carrier wave generation for AM circuits, as a low-frequency clock for microcontrollers (such as 32kHz, commonly used in many microcontroller applications), and, due to its low harmonic distortion, especially in testing and implementation of audio applications, such as in a guitar effects pedal. Furthermore, it can be used to generate an AC source using a DC source.

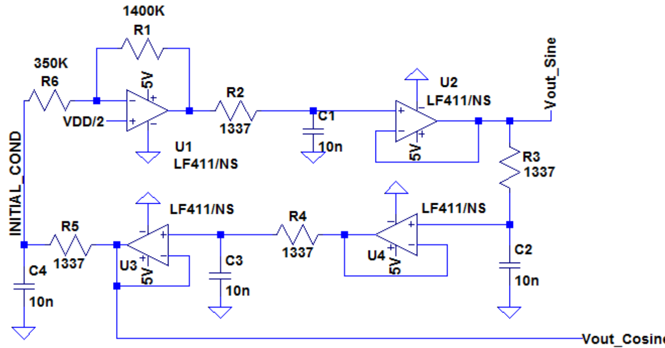
Note that when power is a concern, or when only one voltage supply is available, it is best to utilize the unipolar design. However, when a lower THD and higher output voltage is required, the bipolar design should be considered.

## Tips

### Simulating Oscillation Circuits with SPICE

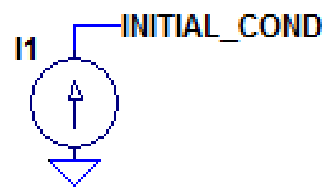
When simulating oscillator circuits in SPICE, it is often necessary to inject an initial condition into the simulation. In the real world, oscillator circuits utilize the noise inherent to the circuit elements to get the initial oscillations started. However, SPICE does not account for this noise in transient simulations, and some models may not provide accurate noise information. As a result, transient simulations of oscillator circuits will often output a steady DC voltage. In order to work around this situation, it is common to inject an initial condition or to use a pulse to make the circuit oscillate. [[Figure whatever]] shows the circuit used in the provided example, along with an initial condition net (INITIAL_COND) attached to the circuit.

**Figure 10.** Example Circuit with INITIAL_COND net attached along the left side.




The initial condition net is attached to a current source, as shown in Figure 11.

**Figure 11.** Example of the initial current pulse to start the oscillation circuit when performing SPICE transient analysis.




This current source is set up to inject a pulse of 20 mA for 0.4 milliseconds at the start of the simulation by using the PWL directive for the current source. This current injection is more than enough to get the simulation started; even briefer amounts of time would work. Equation 3 shows what the PWL specified by the simulation. The values alternate between times (0.0001, 0.0005) and the current output at those times (0.02, 0).

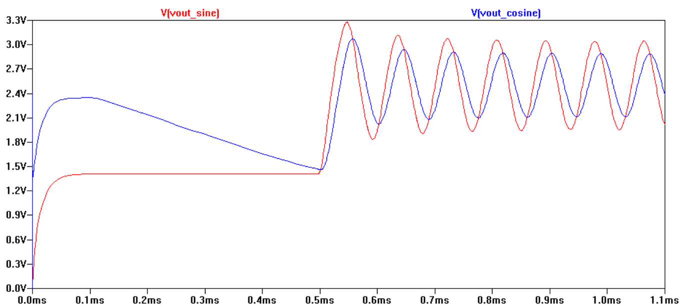
**Equation 3.** PWL definition in SPICE, in order to produce the startup current required to simulate the oscillator circuit. The result is to produce 20 mA of current injection for 0.4ms, starting 0.1ms after the simulation begins.

PWL(.0001 .02 .0005 0)

Additional methods also exist, such as creating an initial condition for a capacitor. Various SPICE tutorials should detail how to do such a thing, if that route is desired over a current injection pulse.

The result of this injection can be seen in Figure 12 below, which demonstrates the startup of the oscillator.

**Figure 12.** Plot showing the use of the initial current injection pulse. This pulse is used to provide sufficient means to start the oscillation.




Note that oscillation does not occur while the current injection is taking place, but begins after the injection ends at 0.5ms. Some amplitude variance occurs before the circuit settles at its normal operational levels.

## Measuring Frequency with SPICE

We can verify the frequency of the waveform by two means: SPICE measurement directives and the Fast Fourier Transform (FFT). The FFT method shown in the Total Harmonic Distortion section above is good for quick visual analysis, but to get the precise frequency, SPICE can be used to take period measurements and calculate a frequency. The .MEAS directive can be used to record data points and to make calculations; results are often displayed in the SPICE output log after the simulation is run.

Equation 4 shows how to use the SPICE directive to calculate frequency.

**Equation 4.** Example using the SPICE .MEAS directive to calculate the frequency of the output voltage.

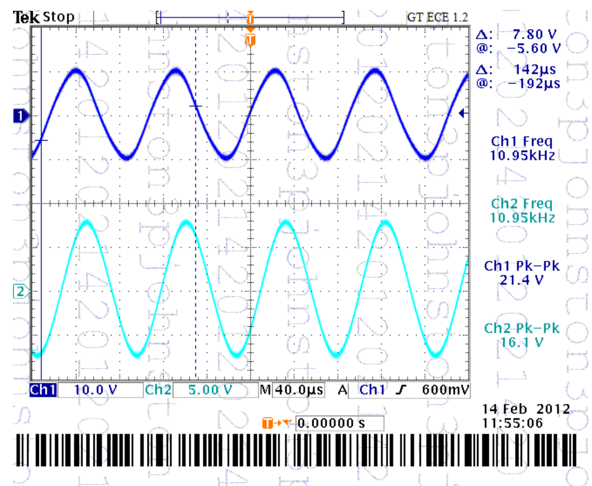
```
.MEAS TRAN t2 V(Vout_Sine) WHEN
V(Vout_Sine) = 2.8 rise = 100

.MEAS TRAN t1 V(Vout_Sine) WHEN
V(Vout_Sine) = 2.8 rise = 99

.MEAS TRAN freq1 PARAM 1/(t2-t1)
```

The first line of the directive tells SPICE to take the time measurement when the sine voltage is above 2.8V. The `rise=100` portion says to use the 100th count of this condition; the 100th count is used to allow the waveform time to settle into the normal operating conditions. The second line does likewise, and says to use the 99th count. Finally, the third line makes the frequency calculation using the two collected data points.

## Malformed Waveforms

If the waveform appears to have the general characteristics of a sine wave, but appears to be distorted or have unequal rise and fall times, check the wiring of resistors and capacitors. Additionally, make sure that the input of the non-inverting terminal of op amp U1 is connected to the correct voltage, VDD/2.

**Figure 13.** Example malformed waveform. This circuit had the incorrect voltage attached to the non-inverting input of op amp U1 when assembling the bipolar circuit for testing.




If the frequency that you observe does not match what the design should provide, check the tolerance on the parts you are using. As typical prototyping labs and hobbyist stocks will utilize 5% resistors and 20% capacitors, there is a large possibility that the design frequency will not be met perfectly by utilizing parts with such high variance. Measuring values with an RCL meter to cherry pick components is a possibility, as well as utilizing potentiometers in series with resistors to adjust the value until the precise frequency can be achieved. However, if a high amount of precision is needed, a true function generator should be used instead of a Bubba oscillator.

## Summary

While function generators may be inaccessible or outside of the budget of many hobbyists or students, there are still methods available to generate the signals needed for testing or for use in various designs. The Bubba oscillator grants the ability to create a sine or cosine wave using components that many students and hobbyists already have on hand, and, if not, that can be acquired for a couple of dollars. Using the appropriate components, it is possibly to generate signals at any desired frequency.

The Bubba oscillator also has the flexibility of being able to operate while using both unipolar and bipolar power supplies.

Aside from use as a "function generator," the Bubba oscillator can also be used for clocking, generating an AC source, from a DC source, and for utilization in high-fidelity audio circuits. This versatile and easily designed circuit can serve many prototyping needs.

## Additional Readings

For more information on the Bubba oscillator, and for examples of other phase shift oscillators, refer to the following sources:


1. Doucet, et al., "DC/AC Pure Sine Wave Inverter," NECAMSID. Available: Worcester Polytechnic Institute, http://www.wpi.edu/Pubs/E-project/Available/E-project-042507-092653/unrestricted/MQP_D_1_2.pdf

2. Mancini, Ron, "Design of op amp sine wave oscillators," Analog Applications Journal, August 2000, pp. 33-37. Available: Texas Instruments, http://www.ti.com/sc/docs/apps/msp/journal/aug2000/aug_07.pdf