Yana Calibrators

Tom Berger K1TRB 11/27/17

There appear to be two calibrators for the AD8307. One is a square wave 5 volt calibrator due to Bob Kopski K3NHI. His first version (QEX Jan/Feb 2004 p. 51) set calibration at -20dbm. More recent versions of the AD8307 do not calibrate well at this power level, so he updated this calibrator to -10dbm. (QEX, Tech Topics, May/June 2010, p. 44). This calibrator is based upon a CMOS 5 volt crystal clock.

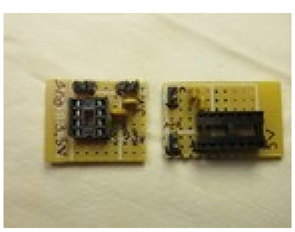
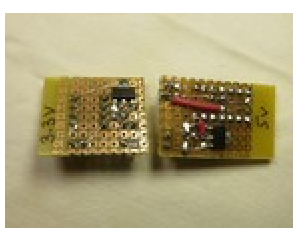
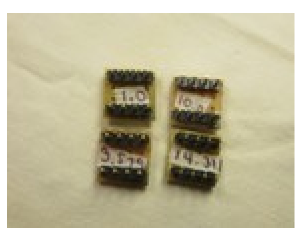
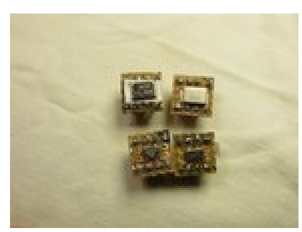
A square wave is the sum of sine waves of all odd multiples of the fundamental. Therefore, it is easy to distort this signal leading to erroneous power level measurements. Todd Gale VE7BPO in his QRP Homebuilder notes (NT7S download, nt7s.com/files/QRPHomebuilder.pdf) describes the Kopski calibrator and gives a sine wave calibrator running from a 12 volt supply. But the sine wave calibrator must be, itself, calibrated. First use a square wave calibrator to set the AD8307 power meter. Then use the meter to calibrate the sine wave calibrator. This bootstrap method accumulates error with each recalibration, but the resulting calibrator is much less subject to signal distortion.

Both of these calibrators are investigated by Rudi Reuter DL5FA with modeling, measurements, and build instructions. http://www.rudiswiki.de/wiki9/RFCalibratorSineSquareWave?highlight=%28%5CbCategoryAmateurRadio%5Cb%29

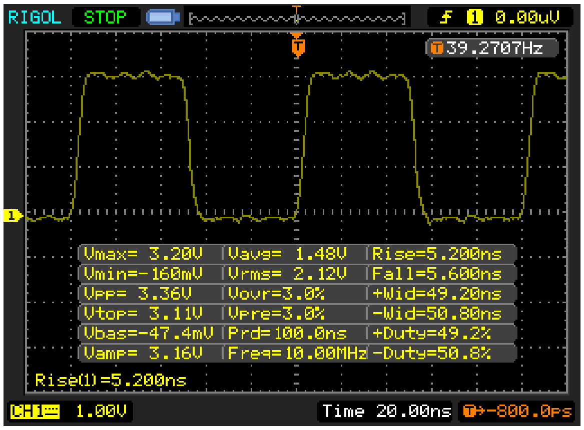
Recently, inexpensive CMOS 3.3 volt crystal clocks have appeared on eBay.
https://www.ebay.com/itm/5PCS-10M-10-000M-10MHz-10-000MHz-OSC-Active-Crystal-Oscillator-0705-7mm-5mm-/251548077854?hash=item3a916f0f1e:g:wi4AAOxyF0pTjXrx
These notes describe two calibrators (square and sine wave) using these clocks. Order several of the 5 x 7mm 10MHz size. I fried one and flipped another into the 4^{th} dimension in my shack.

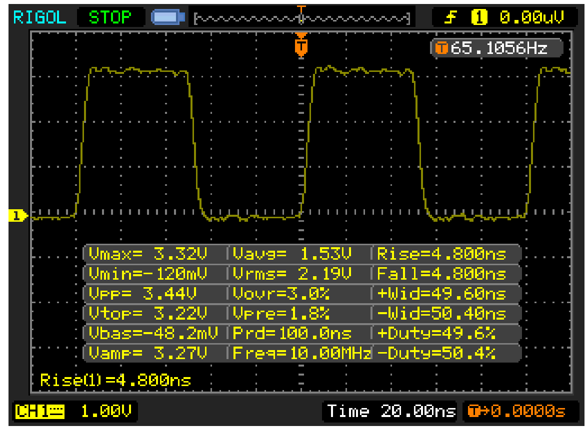
The first calibrator uses a simple square wave signal like the Kopski calibrator. The second uses a lowpass filter to obtain a sine wave calibrator. It would be nice to know the results of builders of the sine wave calibrator to find out if this one can be set with a DC voltmeter. Mine is easily calibrated with a DC voltmeter. If the calibration is repeatable, then this will give a sine wave calibrator which does not need a square wave calibrator to set its accuracy.

A Test Oscillator

These inexpensive crystal clocks present an opportunity to have crystal clocks of varying frequencies. Look below at the Square Wave Calibrator Schematic. Remove all circuitry from the 200 ohm potentiometer onward to the right. Take output directly from the crystal clock. This is the circuit of a simple 3.3V crystal oscillator. Put an 8 pin DIP socket in the location of the crystal and mount various crystals on pin headers. The result is a test module that can accept a wide number of 3.3V crystal clocks. Change the socket to a 16 pin DIP socket and the regulator to 5V, then full and half size oscillators will plug directly into the resulting circuit. The the first two pictures show the 3.3V and 5V test oscillator modules. The second two pictures above show various crystal clocks mounted on pin headers. Some are 3mm X 5mm and some are 5mm X 7mm. I recommend the 5mm X 7mm models..





500 ohm Load

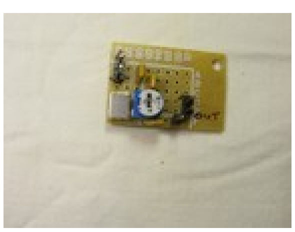
1K ohm Load

The pictures above show the output of the 10.0MHz 3.3V clock mounted in the module. It would be good to know the output impedance of the 3.3V 10.0MHz crystal clock. The two pictures above show the clock loaded first with 500 ohms then with 1K ohms. The output impedance can be calculated from:

$$\frac{Vamp\,500 - Vamp\,1\,K}{Vamp\,1\,K/2 - Vamp\,500} = \frac{3.16 - 3.27}{3.27/2 - 3.16} = 36\,ohms$$

The upper right hand corner frequencies in the pictures are wrong since the trigger point is set incorrectly. Howeve the data calculation gets the frequency right. The value of 36 ohms is used when computing the load resistors in the calibrators.

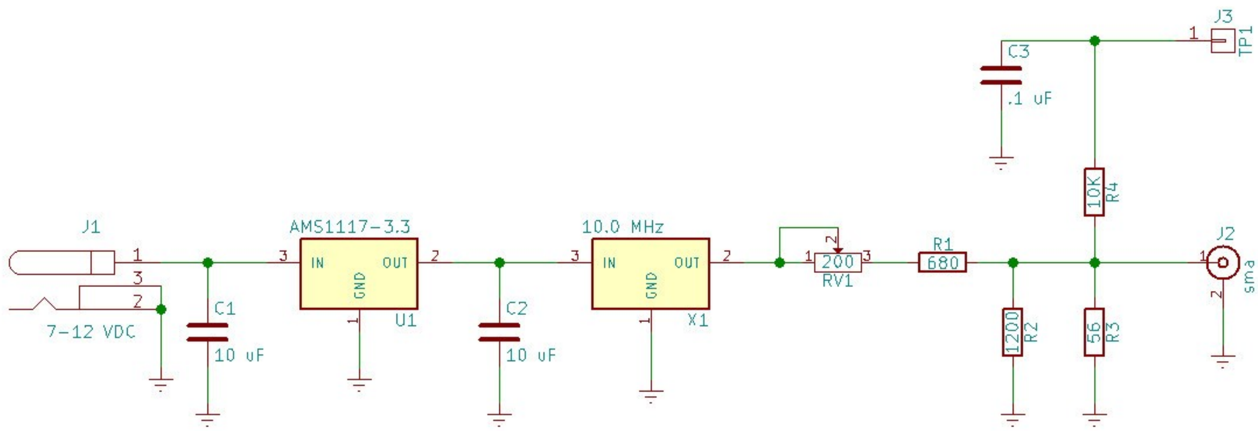
A 10.0 Mhz Square Wave Crystal Calibrator



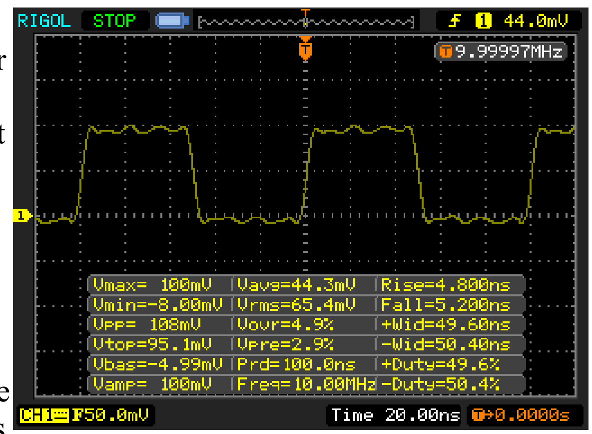
The 5mm X 7mm crystals were selected because they were easier to use. Short lengths of bare wire (30 awg) were soldered to the four corner contacts of the crystal clock. Place the crystal clock with the four contacts up. Hold the crystal down witha weight (like long nose pliers. Tin the four corners and tack a 1 inch wire to each of the four corners.

Setting the crystal upright with writing horizontal, Pin 1 is in the lower left corner. Pin numbers proceed counterclockwise around the crystal. Pin 1 has no connection (used as an anchor point), Pin 2 is ground, Pin 3 is output, and Pin 4 is 3.3V DC. The circuit was layed out on perfboard. I used 0805 smd resistors. The capacitors are leaded. All leaded components should work equally well.

The circuit is the same as the Kopski calibrator: only resistances are changed to account for the lower voltage square wave. The crystal clock and the resistors following it are part of the load presented at the output. The parallel resistors (R2, R3) compensate for this. The square wave at the junction of R1, R2, R3, and R4 must be the same as in the K3NHI calibrator. Therefore, the DC voltage measured at TP1 is the same as for the K3NHI calibrator, namely, 50mV. (Schematic by Kicad)



Square Wave Calibrator Schematic



The picture at the right shows the output of the calibrator when attached to a 50 ohm resistive load. Yana's detector has a blocking capacitor at the input so that it does not present a 50 load ohm at DC. Therefore, Yana should not be used as the load when setting the calibrator. After calibration, the calibrator willl work fine to calibrate Yana. I used a better trigger setting on this picture so the upper right frequency is correct.

The clock must be connected directly to Yana with the shortest possible connection to avoid distorting the square wave. My calibrator has female sma output and Yana has female sma input so I use a male to male sma adapter to connect the calibrator to Yana.

How does it work?

The various sources are, in my opinion, weak in describing why the DC test point is sent at 50mV for correct calibration. First note that this is DC calibration of an AC wave. I consider this a brilliant observation by K3NHI.

Here's how it goes. The oscillator is jumping back and forth between 0V and 3.3V at a 10MHz rate. This means the 3.3V peak to peak square wave represents an average 3.3/2 = 1.65V DC voltage. If we scale this voltage down then the peak to peak square wave voltage is scaled down the same amount. That is, *the DC test point in the circuit tracks the AC wave*. The capacitor C3 and resistor R4 average the voltage of the square wave which is (peak to peak voltage)/2.

So what voltage do we want? We want the peak to peak voltage of the AC square wave to be measured as -10dbm by the AD8307 in Yana. Those who just take this information and do a simple power calculation will get the wrong answer for a surprising reason.

The details are hidden in obscure language in the AD8307 datasheet. First, the AD8307 does not measure power. It measures voltage. For us the conversion from power to voltage occurs in the 50 ohm load at the input to the AD8307. This fact about voltage is hinted at in the labeling of the graph in Figure 10 of the datasheet. In the LOG AMP THEORY section of the datasheet, all calculations are done in voltages. In this same section, the sentence right after equation (2) delivers the punch line. "That is, the incremental gain is inversely proportional to the instantaneous value of the input voltage." And just before the section PROGRESSIVE COMPRESSION comes "... log amps manifestly do not respond to power ... , but rather to input voltage." So we should focus on voltage.

But what voltage? For an AC wave there is peak to peak voltage, peak voltage, average voltage, and rms voltage. To compute power in a wave we use rms voltage. But the AD8307 does not respond to power. The correct choices are either peak to peak or peak voltage (note those words "instantaneous value" in the quotation above).

Maybe you've built a RF diode probe. The usual circuit has a capacitor fed by the diode that charges to the peak voltage. Normally we assume a sine wave, so we multiply the peak voltage by 0.7071 to obtain the rms voltage. This is a peak detecting, rms displaying probe. In fact, most inexpensive ac voltmeters work this way: peak detecting, rms displaying. This is the key to calculations with the AD8307. Different waves have different factors to compute rms from peak. This factor is called the "*Crest Factor*" of the wave and is covered in Wikipedia. There is a nice table on the crest factor page of Wikipedia that gives the value 0.7071 for the sine wave and 1.0 for the square wave (plus a bunch more wave crest factors).

Here is the calculation of power into 50 ohms:

$$-10\,dbm = 0.1\,mW = 0.0001\,W = \frac{Vrms_{sine}^2}{50\,ohms} = \frac{(Vpeak_{sine}\times\sqrt{2})^2}{50} = \frac{Vpeak^2\times 2}{50} = \frac{Vpeak_{square}^2\times 1\times 2}{50} = \frac{Vpeak_{square}^2}{25}$$

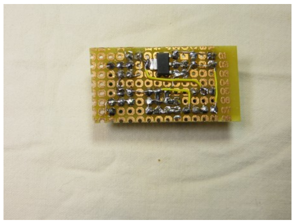
Once the peak voltage is known, we can switch to other voltages so long as we adjust by the crest factor (which is 1 for a square wave). So we get

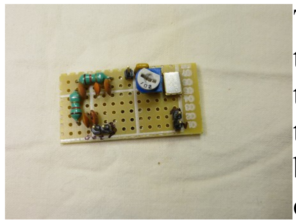
$$Vpeak_{square}^2 = 0.0025 \quad \text{or} \quad Vpeak_{square} = .05\,V = 50\,mV \quad .$$

The peak square wave voltage should be 50mV. But the peak square wave voltage is the same as the average voltage. Since the voltage is varying between 0 and the peak to peak voltage, the average voltage is the DC voltage measured at the test point.

All of this strangeness occurs because the AD8307 is measuring voltage, not power and we are using a square, not a sine wave.

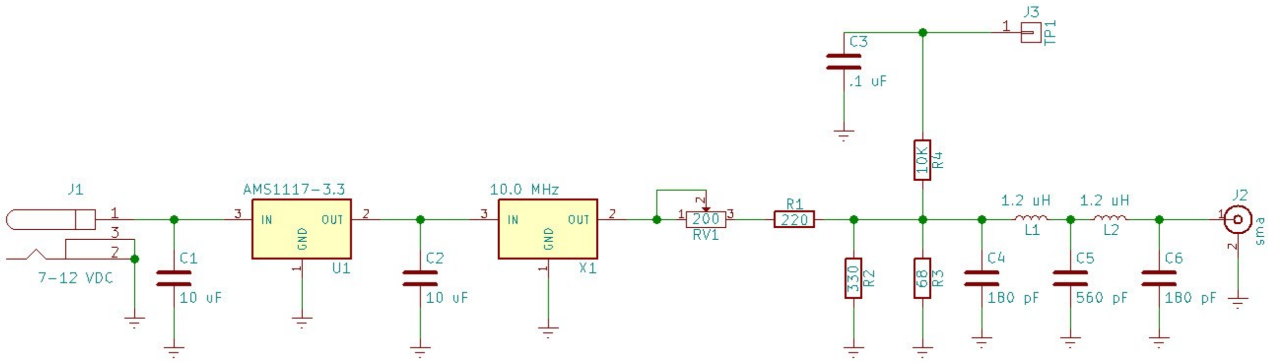
A 10.0 Mhz Sine Wave Crystal Calibrator





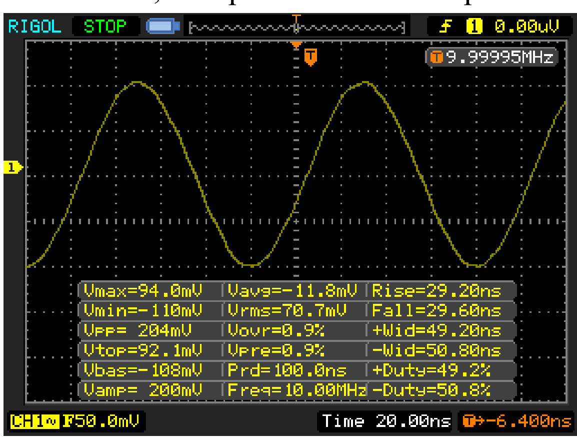
To obtain a sine wave, the square wave is passed through an 11MHz low pass filter. This filter removes the harmonics and lets a 10 Mhz sine wave pass through. The third haromonic (30MHz) is about 25db below the fundamental. This is good enough to calibrate Yana.

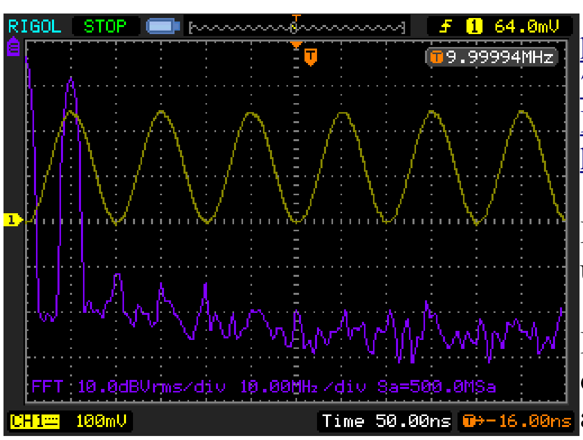
There is loss through the filter so the resistors prior to the filter must change. The circuit shows the revised version adding the low pass filter. The pictures show the top and bottom of the perfboard. The layout of the first part of the circuit is the same as for the square wave calibrator. Leaded components are used for the filter and smd for the crystal clock and regulator.



As for the square wave calibrator, use a 50 ohm load, not Yana, to calibrate the calibrator. For my calibrator, the correct test point DC voltage is 100 mV.

The oscilloscope trace shows the output when loaded with 50 ohms. If you are worried about haromonics, the spectrum of the output is also shown. I set the test point at 100 mV (actually 103 mV).





I wonder if I was just lucky or if other people find the same calibration value. If enough people say, "Yes, that's right." Then there will be no need to build the square wave version to calibrate the sine wave version.

Selecting components for the filter

Turn next to selection of filter components. I purchased 50pcs of 1.2uH coils for $1.58 (about the same price as buying 1 coil). I measured 10 of these and found them all very close to 1.2uH. Therefore, pick one and call it your 1.2uH standard. https://www.ebay.com/itm/50Pcs-0410-1uH-to-1mH-to-4-7mH-DIP-Color-Wheel-Inductor-10-4-7-470-Inductance/192111984263?hash=item2cbac43e87:m:mIUJqGO_61Push9FnNLZG5g

If you have made the LC measuring adapter, it can be used in the following measurments.

If you have a precision LC meter such as a LC meter IIB or a LC100, the design capacitance and inductance values are:

C4(pF)	L1(uH)	C5(pF)	L2(uH)	C6(pF)
178.832	1.1705	578.745	1.1705	178.832

Without such a meter, proceed as follows. Get a collection of 560pF and 180pF ceramic disk capacitors. Working first with the collection of 180pF capacitors set up Yana in SNA mode. Put a 180pF capacitor in parallel with the standard 1.2uH inductor. Set the scan from 9.5MHz to 11.5MHz and set the right marker to 11MHz. Send Yana's signal through the parallel circuit. By selecting capacitors, get the Yana dip close to the green marker. The bottom of the dip should be pretty close to the marker. If all frequencies are high, put a small capacitor in parallel with the 180pF capacitor to bring it close. This parallel combination is your standard 180pF capacitor. If all the measurements are low, switch to 150pF capacitors and use a small capacitor in parallel to pad the measurment to 180pF. Write down the minimum frequency in Yana's tuning mode. Repeat this process to obtain a second 180pF capacitor (parallel combination). The two minimum frequencies should be close.

Using the standard 180pF capacitor, select a second 1.2uH inductor so that the minimum frequency in parallel with the standard 180pF capacitor is very close to the minimum frequency you noted above.

Go back to the standard 1.2uH inductors and put a 560pF capacitor in parallel with it. This time Yana's scan is set from 5MHz to 7MHz and a marker at 6.115MHz. Use the same method as above to select a 560pF combination with a dip close to the green marker.

At this point you have all the parts of the filter: two 1.2uH inductors, two 180pF capacitors, and a 560pF capacitor. The tolerance on your selected values is close enough to make a good filter.

Without this selection process you will obtain a working filter, but it's calibration value may be very different from mine.