Issue #4 - Summer 2004

AmQRP.ORG HOMEBREWER

JOURNAL OF THE AMERICAN QRP CLUB

Featuring ...

"IQ-VFO"

High-Performance DDS-based VFO

by AA0ZZ

Also included ...

RF Power Meter
SPICE Tutorial
PIC-Wx APRS
Morse Sounder
Harnessing Harmonics
Etching PCBs at Home
BLT Tuner Construction
Nuts & Bolts RF Design
... and more!

For electronic builders, experimenters, radio operators, and low power enthusiasts everywhere.

www.amqrp.org

ISSUE #4 CONTENTS

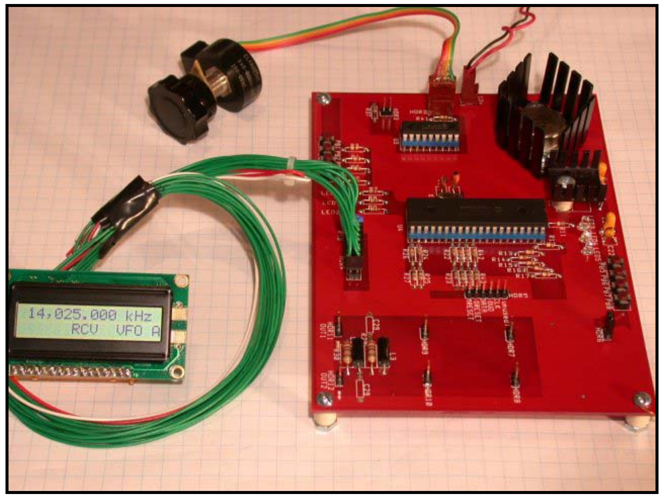
Back to CD-ROM

HOMEBREWER Magazine is a quarterly journal of the American QRP Club, published on CD-ROM. Each issue typically contains over 200 pages of QRP-related homebrewing construction and technical articles intended for builders, experimenters, ham radio operators and low power enthusiasts all around the world. HOMEBREWER features include construction projects for beginners all the way up to the advanced digital and RF experimenters. Annual subscriptions are $15 (for US & Canada) and $20 (for foreign addresses). For information, contact editor/publisher George Heron, N2APB at n2apb@amqrp.org or visit HOMEBREWER Magazine home page at www.amqrp.org/homebrewer.

AmQRP.ORG HOMEBREWER

Craig Johnson, AA0ZZ

IQ-VFO ... a High-Performance HF VFO using the AD9854 Direct Digital Synthesizer



Back in 1997, radio frequency homebrewers really began taking advantage of the conveniences that DDS technology offered: 1-chip simplicity, DC-to-30 MHz signal generation ability and relatively decent signal purity. Since then, AA0ZZ and others have been creating and refining signal generators using this technology. Within the last 12 months, however, the author began working with one of the latest DDS chips from Analog Devices, with an end goal of designing a 0-60 mHz VFO for use with the recently introduced KK7B high-performance R2PRO receiver. This the AA0ZZ story of taming the AD9854 dual-output (I-Q) DDS chip and creating a PCB in the process that others can also use. Additionally, your friendly QHB editor has documented his enclosure for the project, creating a modular high-end VFO component for use on the bench. AA0ZZ and the NJQRP are also making his pc board and PICs available to help others construct this high-performance HF VFO.

Introduction

In the July, 1997 issue of QEX magazine, Curtis Pruess, WB2V published an article featuring a signal generator which used an Analog Devices AD9850 DDS device controlled by a PIC16F84 microcontroller. The signal generator could be used as a VFO, of course. Those radios that need opposite sideband suppression could use this signal generator along with some type of hardware phase-shifting techniques. This project generated a lot of interest and many spin-off projects.

Since that time, a newer device has been introduced by Analog Devices – the AD9854. This device has some significant advantages over the AD9850.

First, the AD9854 produces two outputs with a 90-degree phase difference. This phase relationship remains constant regardless of the frequency. When used in a DC phasing HF receiver or a single-sideband transmitter, these two outputs make it easy to provide opposite sideband suppression over a wide frequency range. The AD9854 data sheet advertises a 0.2 degree maximum phase error.

Second, the AD9854 has a 12 Bit DAC (D/A converter) while the AD9850 has a 10 bit DAC. This is important in reducing the undesirable byproducts (spurs). Dynamic performance (Spurious Free Dynamic Range - SFDR) improves by 6 dB with each additional bit. (See Analog Devices Application Note AN-282.) As a result, the AD9854 has an 80 dB SFDR.

Third, the version of the AD9854 used in this project (the AD9854AST – not the thermally enhanced AD9854ASQ) has a 200 MHz maximum clock speed, compared to 125 MHz for the AD9850. The faster clock speed raises the Nyquist limit and makes the frequency range of 0 to 30 MHz operation easier to attain.

This project is an extension of the concepts introduced in that original AD9850 project, but there have been many major changes. It still has an optical encoder for frequency selection and it still uses an LCD for displaying the frequency. However, the HF-VFO has been changed in these ways:

1) The HF-VFO now uses two PIC microcontrollers - one controlling the optical encoder and the other controlling the AD9854, the LCD, the pushbuttons and the LEDs.

2) The HF-VFO now uses a two-line LCD so more information is displayed.

3) The HF-VFO now operates as two VFOs with two frequencies.

4) The two-part VFO supports split-frequency operation.

5) The two AD9854 outputs are amplified to produce signal levels that are compatible with common receiver requirements (such as the R2 or R2Pro by Rick Campbell - KK7B.)

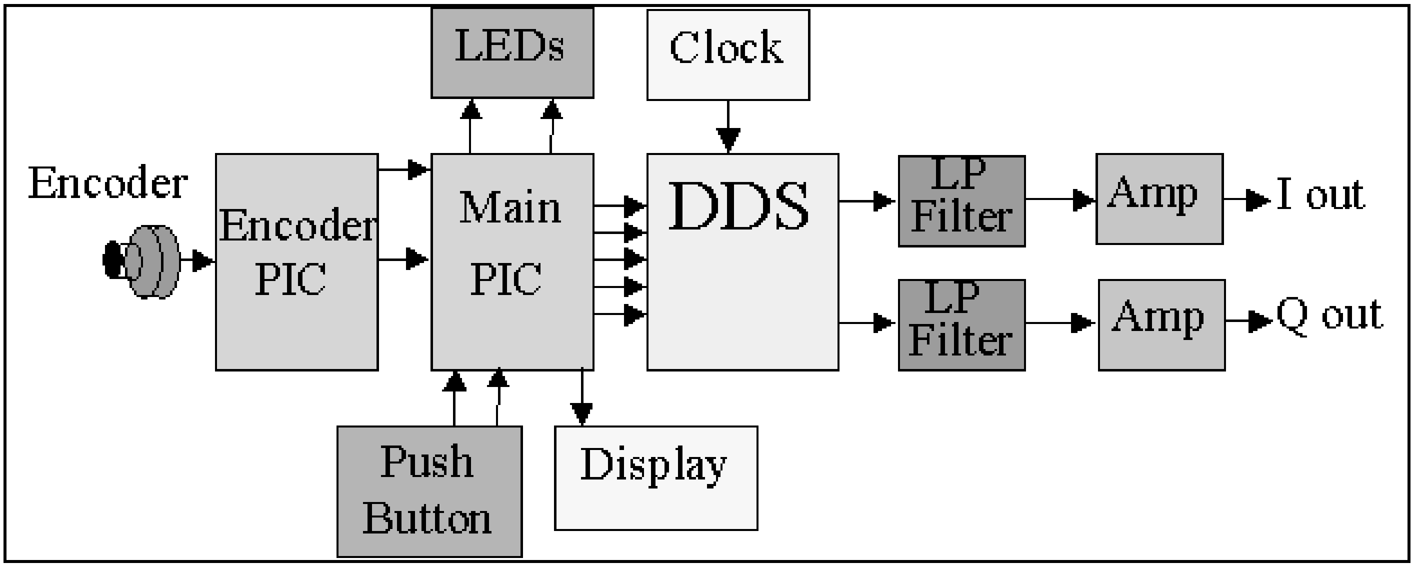


Figure 1: Block diagram of the IQ-VFO

This article describes the AD9854 HF-VFO project that Bruce Stough, AAØED, and I have developed.

Optical Encoder and PIC16F628

Early versions of this HF-VFO project used a single PIC microcontroller for all VFO functions. However, the PIC's main code execution loop was very large and it varied a great deal in length, depending on if the LCD and DDS needed to be updated and if pushbuttons are being used. Since this single PIC also monitored the input signals from the optical encoder, reading the "gray code" to determine how fast the encoder was being turned, a variable size main execution loop made accurate timing and smooth operation very difficult to accomplish.

In this version we decided to use a dedicated PIC16F628 microcontroller for the optical encoder. Why? They are inexpensive and provide a rather elegant solution to the problem of accurate timing for smooth tuning operation. Note that the PIC16F628 is a rather new incarnation of the PIC16F84, with some significant advantages. In this particular application, one of the advantages is that we do not need an external crystal to give accurate operation at 4 MHz. By using a dedicated PIC microcontroller for the optical encoder, the "big brother" PIC16F877 microcontroller can easily handle the other operations it is called upon to perform.

The encoder PIC monitors the output of the encoder. Periodically it sends information to the main ('877) microcontroller to indicate that the encoder tuning knob has been turned as well as the direction of change. The encoder ('628) microcontroller sends the data to the '877 PIC by setting up 3 data lines and a direction line and then generating a pulse on the '877 PIC's interrupt line. The interrupt causes the '877 PIC to stop its current operation for a very brief period of time, to examine the data lines, save the new magnitude and direction, set a flag saying a change has been made, and then return to the task it was executing at the time of the interrupt. A very short time later, code in the main execution loop will detect the "change" flag and make the appropriate frequency change in the DDS and LCD.

The amount of change in the frequency is determined by an interesting algorithm. The optical encoder that we use puts out 128 pulses per revolution. Thus we have 256 transitions for each of the two output lines for a total of 512 transitions per revolution.

We decided we want to have a maximum of 20 tuning updates per second. 20 updates per second converts to 50 ms per update. We want to count the number of transitions in 50 ms and send that number to the ‘877 PIC. We can easily handle this rate of interrupts in the ‘877 PIC without having to stack them or lose them.

We also decided we want the PIC to be able to capture all transition pulses when we turn the tuning knob (optical encoder) at a rate of least 2 revolutions per second. (When we turn the knob faster than 2 revolutions per second we will allow some transitions to be skipped.) At this rate, the maximum number of transitions per update would be:

512 transitions/rev * 2 rev/second = 1024 transitions/sec.

1024 transitions/sec = 0.98 ms / transition

then:

50 ms/update divided by 0.98 ms/transition = 51 transitions/update

We could send this number over with 6 "binary" bits (63 max). However, we have chosen to use this alternate scheme to do it with 3 "binary" bits:

The encoder PIC counts the number of transitions in 50 ms. It sends over an "accelerated" value by looking up a value in a table. The value ENCINDEX in the following table (represented as 3 bits of "data sent") is sent to the ‘877 PIC microcontroller. Then the ‘877 PIC determines the actual size of the frequency step, depending on whether the current tuning rate is slow or fast.

Transitions in 50ms	**Enc**	**Data**	**FSTEP SLOW**	**FAST**
1	0	000	1	1
2	1	001	2	2
3-4	2	010	5	10
5-8	3	011	10	60
9-16	4	100	100	600
17-32	5	101	300	1800
33-64	6	110	1000	6000
65+	7	111	5000	30000

In this scheme, since 65 is the largest number of pulses that can be counted and distinguished with 3 data bits, we are limited to a little over 2 revolutions of the encoder per second. (If the encoder is turned faster, and more than 65 pulses occur in the 50 ms window, the result which is sent to the ‘877 PIC is still a "7", the same as it would be if only 65 pulses occurred.)

How well does it work? In slow-tune mode you can tune in 1 Hz increments and accelerate up to about 25 kHz per revolution. In fast-tune mode you can still tune in 1 Hz increments but can accelerate up to about 150 kHz per revolution of the encoder.

Mechanical Encoder instead of Optical Encoder

Since the optical encoder specified for this project is a fairly expensive item (unless you can find it in a "surplus" bin somewhere, as I did), you may want to experiment with a much less expensive alternative – a mechanical encoder. DigiKey has one for about $3.50, instead of about $40 for their optical encoders. The difference is that the mechanical encoder only has 24 positions per revolution, compared to 128 for the optical encoder. The result is that the tuning is not quite as smooth and responsive as with the optical encoder. However, it does work fine.

I have changed the acceleration algorithms to the following when the mechanical encoder (24-position) mode is selected.

Transitions

in FSTEP

50ms	Enc	Data	SLOW	FAST
1	0	000	1	1
2	1	001	5	7
3-4	2	010	25	50
5-8	3	011	50	300
9-16	4	100	500	3000
17-32	5	101	1500	9000
33-64	6	110	5000	30000
65+	7	111	25000	150000

With this algorithm, it is still possible to tune in increments of 1 Hz. However, the acceleration increments change more rapidly than with the optical encoder.

By default, the board is initialized to use the optical encoder acceleration table as the primary and the mechanical encoder table as the alternate. (This priority order can easily be changed, of course.) To select the alternate encoder table, this sequence of button presses is used:

1) Press and hold Pushbutton 1

2) Press and hold Pushbutton 4

3) Release Pushbutton 1

4) Release Pushbutton 4

You can tell whether the primary or alternate encoder table is selected by pressing Pushbutton 7 to select fast-tuning mode. If the character in the far right side of the first line becomes a "+" when in fast-tuning mode, you are using the primary encoder table. If the character becomes a "#’ when in fast-tuning mode, you are using the alternate encoder table.

Once the alternate encoder acceleration table is selected, it remains selected until the VFO is powered down. Furthermore, the encoder selection is not remembered over the power-down; you must enter it again in the next session. Obviously, once you have settled on an encoder, you should change the code to use this selection as the primary encoder.

Main processor – the PIC16F877

The PIC16F877 microcontroller was selected for the main controller for several reasons. First of all, it has more I/O pins than are available on the smaller PIC microcontrollers. This gives us the opportunity to use dedicated pushbuttons for operator input as well as dedicated LEDs for operator feedback, without resorting to complicated multiplexing. We think a high-end radio, which this VFO project certainly can support, is easier to operate when it has dedicated pushbuttons. Single button radios and menus are more appropriate for other applications.

Another significant advantage of using a PIC16F877 microcontroller was for ease of development. We were able to use Microchip’s In-Circuit-Debugging (ICD) tool. This made our development time much more productive and enjoyable. The version of ICD that we use only supports the high end PIC microcontrollers. (The ICD is available from DigiKey. Part number DV164001-ND.)

Pushbuttons

This HF-VFO version has 8 pushbuttons to control the VFO.

Pushbutton 1: 1 MHz UP

Each time this button is pressed the frequency will increase by 1 MHz. Frequency maximum is 30.000 MHz.

Pushbutton 2: 1 MHz Down

Each time this button is pressed the frequency will decrease by 1 MHz. Frequency minimum is 0.000 MHz.

Pushbutton 3: Normal/Fast Tuning Rate

Pressing this pushbutton toggles the optical encoder tuning rate between normal and fast tuning.

The default tuning rate allows fine tuning in 1 Hz steps when the encoder is turned very slowly. When turned slowly, one complete revolution of the encoder shaft will change the frequency by 512 Hz. However, the acceleration algorithm provides greater rate of frequency change when the encoder is turned faster. In normal tuning mode the maximum amount of frequency change in one revolution of the encoder shaft has a practical limit of about 25 kHz.

After toggling the pushbutton to fast tuning mode, the operator can still tune in 1Hz steps by turning the shaft very slowly. However, the acceleration algorithm ramps up much faster, so the maximum tuning rate is about 150 kHz per revolution of the encoder shaft.

When in fast-tuning mode, the right-most character on the first line of the LCD displays

a "+" (plus sign) or a "#" (pound sign). If the indicator is a "+" it means you are in fast-tuning mode and the primary encoder acceleration table is in use. If the indicator is a "#" it means you are in fast-tuning mode and the alternate encoder acceleration table is in use.

See the section regarding the use of a mechanical encoder instead of an optical encoder for more details.

Pushbutton 4: Special Select

This button is used in two special situations.

First, it is used during initial board debug. Since it is very difficult to see some of the control signals being sent to the AD9854, and since some of these essential signals are only generated at AD9854 initialization time, it is often very difficult to see them on an oscilloscope to verify that they are getting to the AD9854. This special debug mode is enabled by holding this pushbutton down while powering up the board. It must be held down for about 2 seconds - until the introduction/debug display on the LCD screen has gone and the normal frequency display is shown.

Second, it is used in conjunction with Pushbutton 5 for selection of the alternate encoder. See details in the section describing the use of a mechanical encoder rather than an optical encoder.

Pushbutton 5: Set VFOA = VFOB

When this button is pressed, the frequency of the VFO that is not currently being displayed will be changed to match the frequency of the VFO that is currently being displayed.

Pushbutton 6: Split Frequency Operation

This pushbutton allows the operator to toggle between NORMAL and SPLIT mode.

SPLIT mode operates in conjunction with the keying interface header on the board.

When this keying circuit is activated (by grounding the key line or shorting the interface pins together) and it is in SPLIT mode, the PIC will switch from one VFO to the other. In NORMAL mode the frequency remains at the displayed frequency (VFO-A or VFO-B) as the keying circuit is activated.

The operator sets desired transmit frequency in one VFO and the desired receive frequency in the other VFO. The current VFO selection is done by means of the Rx VFO Toggle pushbutton. The LCD displays the frequency of the receive VFO when in receive mode and it displays the frequency of the transmit VFO when in transmit mode. The LCD displays the word XMIT when the keying circuit indicates transmit mode and RCV when in receive mode.

The LCD displays the word SPLIT when in SPLIT mode. This portion of the LCD is blank when in NORMAL mode.

Pushbutton 7: Rx VFO Toggle (VFOA / VFOB)

Two VFO frequencies are used in the HF-VFO. This pushbutton allows the operator to toggle between VFO-A and VFO-B as the currently active VFO. The LCD indicates whether VFO-A or VFO-B is active.

Pushbutton 8: Calibrate

The Calibrate button is used to put the VFO into a special mode to ensure the DDS is generating a sine wave for the exact frequency that is displayed on the LCD.

First a bit of background. When we want the AD9854 to generate a sine wave at a particular frequency, we need to send it a 6-byte number referred to as the Frequency Tuning Word. The AD9854 uses the Frequency Tuning Word and the reference clock to generate the output waveform.

The AD9854 can generate an output frequency with 48-bit accuracy. However, in this application, since we are always going to be generating a frequency which is a whole number of Hertz, we keep a 4-byte numeric constant in the PIC's EEPROM which represents a 1 Hertz change in output frequency when used with the specified reference clock. We refer to these 4 bytes as:

ref_osc_3 (the most significant byte)

ref_osc_2 (next byte)

ref_osc_1 (next byte), and

ref_osc_0 (the least significant byte)

Note that the AD9854 has a 48-bit frequency tuning word and we are only saving a 32-bit constant in EEPROM. For performance reasons, and since 48-bit accuracy is not required for this application, we will construct the 48-bit tuning word from the 32-bit value. We will use this value to calculate the 48-bit frequency tuning word. We will then multiply the 32-bit value by the desired output frequency, in Hertz, to obtain another 32-bit product. This 32-bit product will be used as the most significant 32 bits of the 48-bit frequency tuning word and we'll fill the remaining 16 bits with zeros.

This numeric constant is interpreted as a fixed point real number in the format:

<refosc3> <refosc2> <refosc1> <refosc0>

where

refosc3 = (2^32/ osc_freq_in_Hz)

refosc2, refosc1, and refosc0 are the fractional part of:

(2^32 / osc_freq_in_Hz) times 2^24.

For example, for a 125 MHz clock:

refosc3 is (2^32 / 125 x 10^6) = 34.359738368 truncated to 34 (0x22)

refosc2 is the high byte of:

(.359738368 x 2^24) = 6035408.303

6035408.303 = 0x5C17D0, so high byte is 5C.

refosc1 is the next byte of 0x5C17D0, or 17

refosc0 is the last byte of 0x5C17D0, or D0

The values for common oscillator frequencies are as follows:

Frequency	**ref osc3**	**ref osc2**	**ref osc1**	**ref osc1**
125.00 mHz	**0x22**	**0x5C**	**0x17**	**0xD0**
120.00 mHz	**0x23**	**0xCA**	**0x98**	**0xCE**
100.00 mHz	**0x2A**	**0xF3**	**0x1D**	**0xC4**
90.70 mHz	**0x2F**	**0x5A**	**0x82**	**0x7A**
66.66 mHz	**0x40**	**0x6E**	**0x52**	**0xE7**
66.00 mHz	**0x41**	**0x13**	**0x44**	**0x5F**
60.00 mHz	**0x47**	**0x95**	**0x31**	**0x9C**
50.00 mHz	**0x55**	**0xE6**	**0x3B**	**0x88**

Once again, we can think of this constant number as the step size which represents 1 Hz of movement. When we want to set the VFO's frequency to a specific number of Hertz, we simply have to multiply this constant by the desired number of Hertz to obtain a Frequency

Tuning Word. We send this Frequency Tuning Word to the AD9854 and, *voila*, it generates the sine wave at the specified frequency. In fact, it generates two outputs exactly 90 degrees apart.

Now we can see why we need the Calibrate function. When the PIC microcontroller is programmed, we set up preset values that are loaded into the PIC's EEPROM, based on the nominal frequency of the AD9854's external reference clock. However, it is never absolutely accurate as specified. The Calibrate function is used to adjust the 4-byte constant (step size) such that when the AD9854 uses it with this reference clock, the output frequency is accurate. The updated 4-byte constant is then stored back in EEPROM for future use in calculating the Frequency Tuning Word. It is not lost when the HF-VFO is powered down.

When the Calibrate button is first pressed, the VFO switches to 10 MHz and the optical encoder tuning rate is set to a very small number. Now we tune a nearby receiver to WWV at 10 MHz and we listen, in CW mode, to WWV's tone at some offset (e.g. 700 Hz above). We also can hear the HF-VFO putting out a faint signal at what "it thinks" is 10 MHz with the same tone offset. While listening to the WWV tone and the HF-VFO tone, we can turn the encoder to adjust the HF-VFO's tone. As the two frequencies get closer together, we can hear the "beats" get slower and slower. We listen for "zero beat" which indicates the tones are identical. When the two tones are identical it is calibrated. We press the Calibrate button again to exit this mode. The updated constant is now saved in EEPROM.

Other Controls

A keying circuit interface header is provided on the board for a connection to external keying circuitry. It is used for switching between the two VFO frequencies when in SPLIT mode. The LCD changes from RCV to XMIT when the keying interface pin is grounded (or the two header pins are connected together). In addition, if SPLIT mode is selected, and the PIC changes to use the opposite VFO frequency when the keying interface pin is activated (grounded).

Test this feature by entering different frequencies in VFO-A and VFO-B. Select SPLIT operation. Now short the two header pins together and watch the display change from one VFO to the other and the display change from RCV to XMIT. By design, the currently selected VFO is always the "receive" VFO, and the receive frequency is displayed. The other VFO contains the "transmit" frequency and it is displayed when transmitting.

LCD Display

The LCD display shows the current frequency that is being generated by the DDS.

It also shows the state of the SPLIT toggle, the current transmit/receive mode, and the currently active VFO (A or B).

Virtually any 16 x 2 LCD which uses a HD44780 controller can be used. Surplus parts suppliers are a good source for more inexpensive parts.

A variable resistor (R10) is included to adjust the LCD brightness. This resistor is not be necessary for all LCDs but may be required. For some types of LCDs, Pin 3 may be tied directly to ground, while in others a small positive voltage is required on Pin 3 for maximum LCD brightness.

LEDs

There are several Light Emitting Diodes on the board. Eventually they will be moved to the front panel.

LED 0 -- This LED for debug purposes. It is very briefly turned on when bit 0, the least-significant bit, is set in the encoder data by the encoder PIC (the 16F628) when an encoder interrupt is generated.

LED 1 -- This LED is for debug purposes. It is turned on during the period when LCD is being initialized. During operation, it is also turned on when bit 1 is set in the encoder data when an encoder interrupt is generated.

LED 2 -- This LED is for debug purposes. It is turned on during the period when the DDS is being initialized. During operation, it is also turned on when bit 2 (the highest-order bit) is set in the encoder data when an encoder interrupt is generated.

LED 3 -- This LED is turned on when VFO-A is active in receive mode.

LED 4 -- This LED is turned on when VFO-B is active in receive mode.

LED 5 -- This LED is turned on when SPLIT operation is active.

Circuit Description

Figure 1 shows the schematic for the HF-VFO.

The two socketed PIC microcontrollers control the HF-VFO functions. The PIC16F628 is a 16-pin part, while the PIC16F877 is a 40-pin part. Each PIC pin that is used as an input has a 1k ohm pull-up resistor attached to +5 v.

The optical encoder feeds its two "gray code" signals to the PIC16F628.

The HF-VFO uses a 125 MHz DIP clock oscillator for the AD9854. This was a tradeoff. We could have used a lower frequency clock oscillator and then used the AD9854’s internal clock multiplier, but this introduces additional phase noise as well.

The output current (Iout) from the AD9854 is set by the size of the Rset resistor. We are using an Rset of 3.9k which translates to 10ma of output current. (20 ma is maximum, but 10 ma is selected because it provides the best SFDR.)

Software Description

We run the AD9854 in serial mode. This means we use a single data line, a clock line, and an external update line to send the frequency updates from the PIC to the AD9854. Two reset lines are also controlled by the PIC.

Determining how to get the AD9854 initialized was a real challenge, especially with a 125 MHz reference clock. The problem stems from the fact that the AD9854 powers up in internal update mode while it really needs to be in external update mode to operate in this type of application. See the AD9854 data sheet for details. After a lot of tinkering we were able to get this working properly.

Board Description

The layout is on a single-sided 4.6" x 6.6" (114 x 165 mm) board. Surface mount parts are mounted on the copper trace side of the board while all other parts are mounted on non-trace side.

I did the board layout using WinBoard from Ivex, with some custom parts such as the ERA-3's, the AD9854, and the surface mount (0805 size) parts.

External power to the board is 12 - 13.5 volts. The ERA-3 bias scheme uses 12 volts. A large TO-3 version regulator is used for the conversion from 12 volts to 5 volts while a TO220 version regulator is used to drop from 5v to 3.3v.

The AD9854 must run at 3.3 volts, so its reference clock (125 MHz) is also a 3.3v part.

However, since the PIC16F877 runs at 5 volts and the AD9854 runs at 3.3 volts, we needed a method of converting the voltages of the signal lines that connect these two parts. We did it with simple voltage dividing resistors on each signal line.

All pull-up resistors on PIC ports are 10K, 1/8 watt, and all surface mount resistors and capacitors are 0805 size.

Power consumption for the entire board is about 425 ma but some of this is "burned up" by the 5-volt regulator (dropping from 12 volts to 5 volts). Output of the 5-volt regulator is about 400 ma. About 375 ma of this goes to the AD9854 and the 125 MHz clock.

Mounting the AD9854

Much has been said about the challenges of working with the AD9854 with its 80-lead LQFP format. The .65 mm pin spacing does provide a challenge but it is not an insurmountable problem for homebrewers. I use a WTCPT soldering iron with a 1/64" tip.

I started by putting solder flux on the entire AD9854 pad area of the board. Then I tinned all the pads by applying liberal amounts of solder. I removed the excess solder by drawing a piece of solder braid, with my soldering iron heating it, across the pads. It doesn't need much solder on the pads.

To mount the AD9854, I used a lighted magnifying glass. I carefully placed the device, making sure pin 1 was in the correct position. I moved it around carefully with a toothpick. When I was confident that all 80 pins were lined up properly on their pads, I tacked one corner by touching the pin briefly with my soldering iron. Once again I examined it carefully. Note that it is a lot easier to remove the part when only one pin is tacked. When all 80 pins are soldered, I think you may have to use a heat gun to remove it. (Thankfully, I haven't had to try this trick!) Satisfied, I tacked another pin in the opposite corner. After checking again very carefully, I proceeded to solder all the pins by simply drawing my soldering iron along the row of pins. It is easy to see the solder flow as the pins attach to the pads.

Now I checked it completely for unsoldered connections and possible shorts. I did this with an ohmmeter with clip leads and a common pin. I clipped one lead to a ground connection point and clipped the other lead around the common pin. One at a time I touched the common pin to each AD9854 pin that was supposed to be grounded and watched the meter to verify the connection. Then I did the same thing for all the Vdd (3.3v) connections. Finally, I verified all the PIC signal lines to the AD9854 by checking for continuity between the interface header pins and the AD9854 pins. Yes, at first I did have a couple of AD9854 pins that were not soldered completely, so I again touched these individual pins with my soldering iron. Then I checked the area again.

*[**NOTE:** Homebrewers who purchase the pc board from the NJQRP will have an option available to have the AD9854 DDS chip soldered on.]*

Enclosure

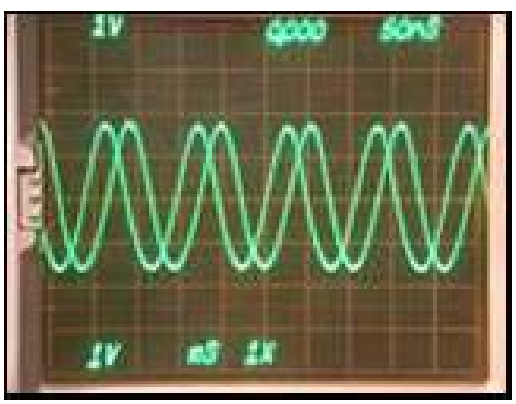
It would be good to mount the HF VFO pc board into a sturdy metal enclosure, both for protection as well as EMI isolation. After all, you wouldn't want all the digital signals causing birdies in rigs that you will be driving with this VFO!

George, N2APB worked closely with me on the packaging of the project and developed an example of a convenient enclosure. See some details and photos at the end of this article.

Performance

Size of outputs

Each of the channels produce an output waveform which is about 2 volts peak-to-peak (10 dBm). See the I & Q output waveforms below.



Opposite sideband rejection

Initial testing by shows that an R2Pro receiver will get at least 40 dB of opposite sideband rejection.

Spurs - TBD

Phase noise - TBD

What's Next?

There are a number of directions to go from here. Here are a few of the areas that I can immediately see:

1) Use another kind of DDS Reference clock to see if that makes the output "cleaner". Some people have said that a DIP clock is not nearly "good enough". Find out. Make an oscillator with a crystal and stabilization circuitry and see it is better. Check the phase noise. See Application Note AN-419 at the Analog Devices web site: http://www.analog.com or AN-263 from National Semiconductor at http://www.national.com/an/AN/AN-263.pdf

2) Add a PLL to the output and observe the reduced spurs at the output. See the trade-off between DDS spurs and the phase noise introduced by the PLL's VCO.

3) Experiment with dithering to reduce spurs on the output. See A Technical Tutorial on Digital Signal Synthesis at the Analog Devices web site. Also see High Speed DAC's and DDS Systems by Walt Kester at the Analog Devices web site. http://www.analog.com

4) Change the MMIC amplifier. Take the signal from the header at the output of the Low Pass Filter to an amplifier of your choice via 50-ohm coax.

5) Change the Low Pass Filter. Try using larger through-hole components.

6) Add an offset for an audible CW sidetone.

7) Change the 5-volt regulator to a 5-volt switching power supply for less heat generation.

8) Make a computer interface for the DDS, instead of using the PICs. The 6-pin header between the 16F877 and the DDS would be an easy connection point; all the essential DDS signals come from that header. You may want to use a parallel port. Then make your computer's operator interface as simple or as elegant as you want. Maybe do your tuning with a mouse or with the keyboard. Maybe even a touch screen.

9) Change the tuning acceleration algorithm to your liking.

10) Use another kind of LCD. Larger/smaller. Display more information.

11) Add code/circuitry for changing the selection of USB/LSB. This selection can be accomplished by simply reversing the I and Q output connections to the external receiver/transmitter. (The I and Q outputs of the AD9854 are always 90 degrees out of phase and there is no way to reverse them by programming in the AD9854.)

12) Add code/circuitry to handle Transmit/Receive timing. We may need to add another PIC for controlling this keying. This would allow an exact adjustment for the delay in switching between Receive and Transmit. (This is different for every transmitter/receiver.) We must be careful to switch from the receive frequency to the transmit frequency before the transmitter starts transmitting, and we must NOT switch from the transmit frequency back to the receive frequency until after the transmitter has stopped transmitting. Currently the main PIC instruction loop takes about 10 us when no LCD/AD9854 updates are being done, and about 1.5 ms when updates are being done. This means that, worst case, the keying change will be detected by the PIC in 1.5 ms and the PIC will change between the transmit frequency and the receive frequency. This is probably too fast for most receivers/transmitters.

13) Add frequency memories. They could be saved in EEPROM. Additional pushbuttons or switches may be required.

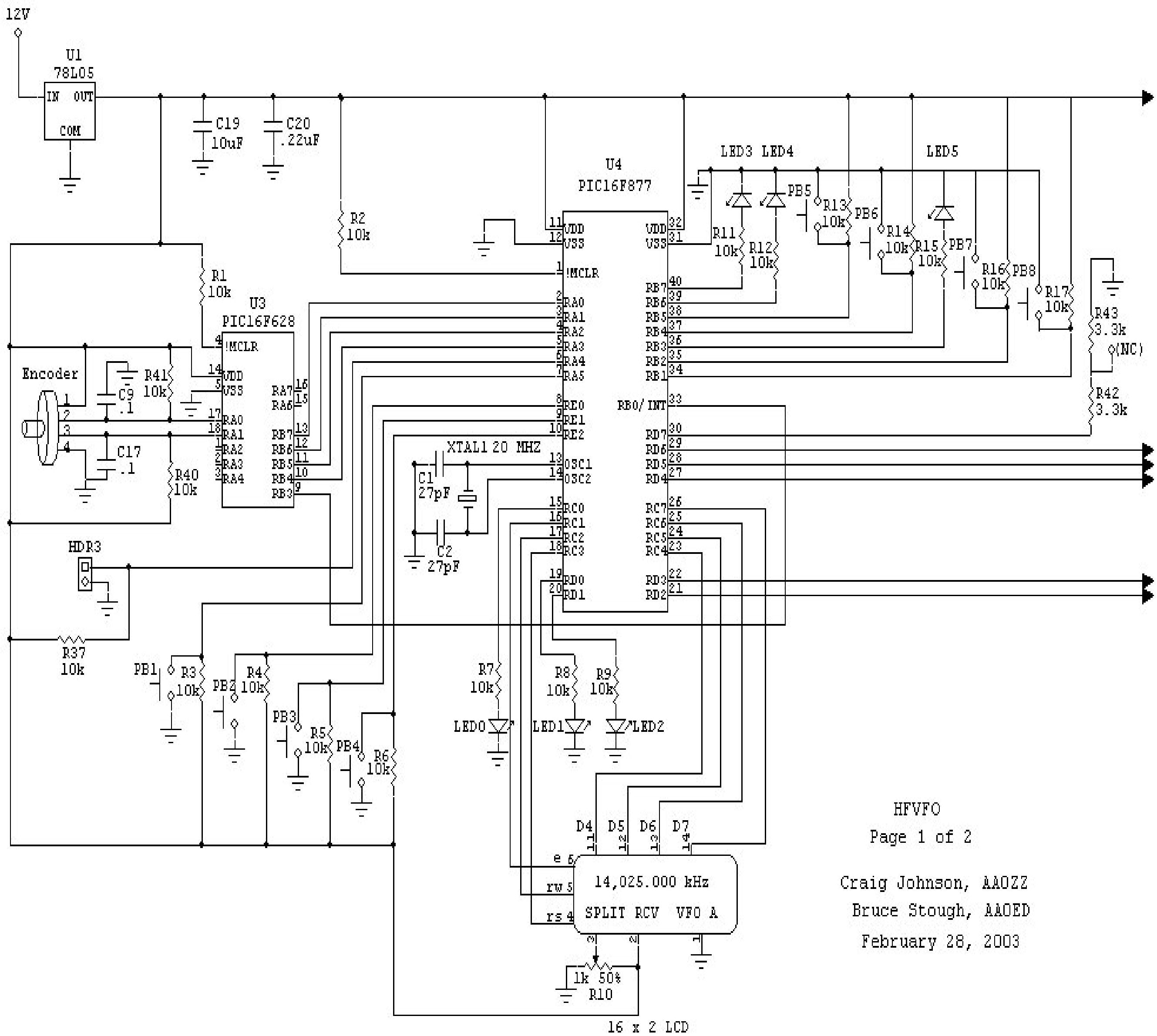
14) Add another PIC processor. Maybe use a dedicated 16F877 for a more complete user interface – more pushbuttons, LEDs, and a maybe a keypad. The new 16F877 could communicate with the "main" 16F877 which controls the DDS and the LCD.

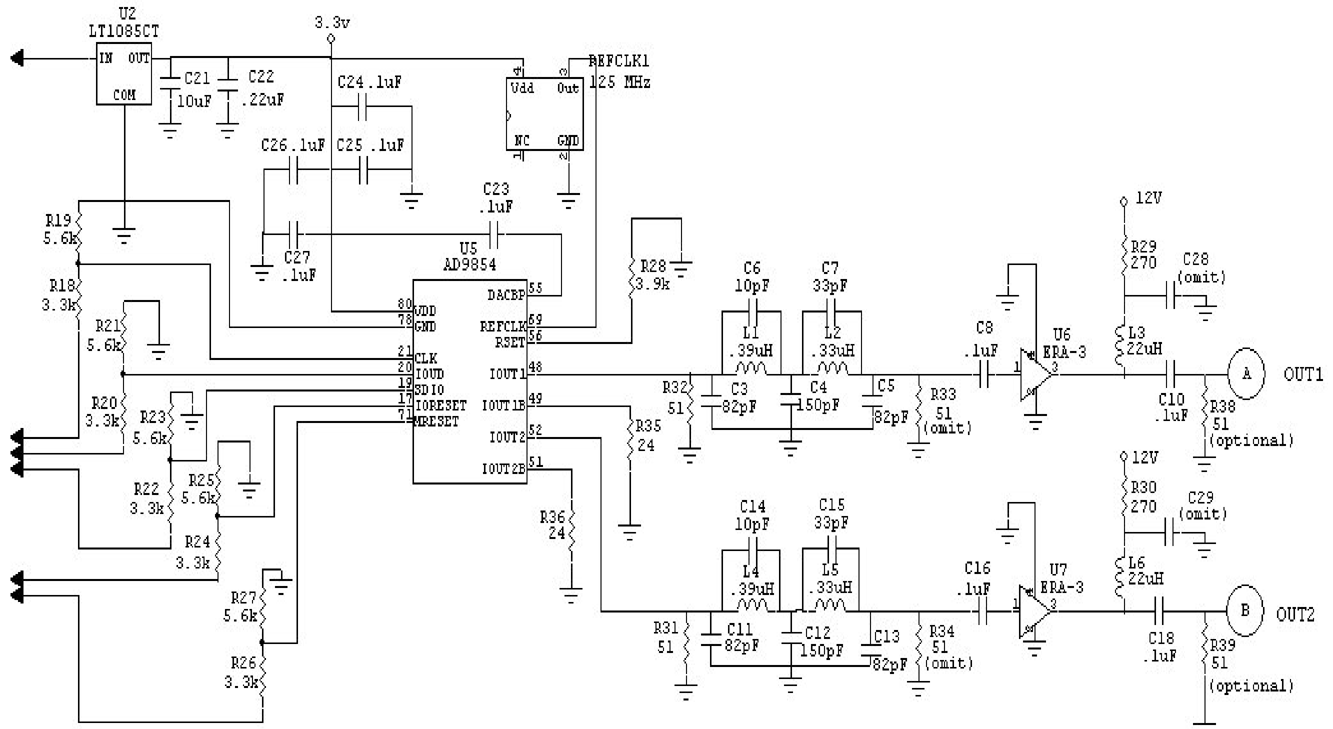
15) Use the HFVFO to experiment with frequency-hopping Spread Spectrum.

Summary

This project is another step in the direction of a usable DDS–based VFO for a high-end radio. It has opened some new doors and has started us thinking about additional features that we would like to implement in future versions. However, it is likely that every user will have unique ideas about what is important to them. We hope this project will be a springboard for others use as they branch off in multiple directions.

To get the very latest version of the various components of this project, visit my web page: http://www.cbjohns.com/aa0zz


12V
U1
78L05
IN OUT
COM
C19
10uF
C20
.22uF
U4
PIC16F877
R2
10k
R1
10k
U3
PIC16F628
Encoder
R41
10k
C9
.1
C17
.1
R40
10k
HDR3
R37
10k
LED3 LED4
LED5
PB5
PB6
PB7
PB8
R11
10k
R12
10k
R13
10k
R14
10k
R15
10k
R16
10k
R17
10k
R43
3.3k
(NC)
R42
3.3k
XTAL1 20 MHZ
C1
27pF
C2
27pF
PB1
R3
10k
PB2
R4
10k
PB3
R5
10k
PB4
R6
10k
R7
10k
R8
10k
R9
10k
LED0
LED1
LED2
D4 D5 D6 D7
14,025.000 kHz
SPLIT RCV VFO A
1k 50%
R10
16 x 2 LCD
HFVFO
Page 1 of 2
Craig Johnson, AA0ZZ
Bruce Stough, AA0ED
February 28, 2003



AD9854 Connections (not all shown)

Gnd: 1-8, 11, 12, 14-16,
18, 22, 26-28, 30,
33-34, 39-40, 41-43,
45-47, 53, 59, 61-62,
64, 66-68, 70, 72, 75-78

VDD: 9-10, 23-25, 31-32,
37-38, 44, 50, 54, 60,
65, 73-74, 79-80

No Connection: 13, 29, 35-36, 57

HFVFO

Page 2 of 2

Craig Johnson, AA0ZZ

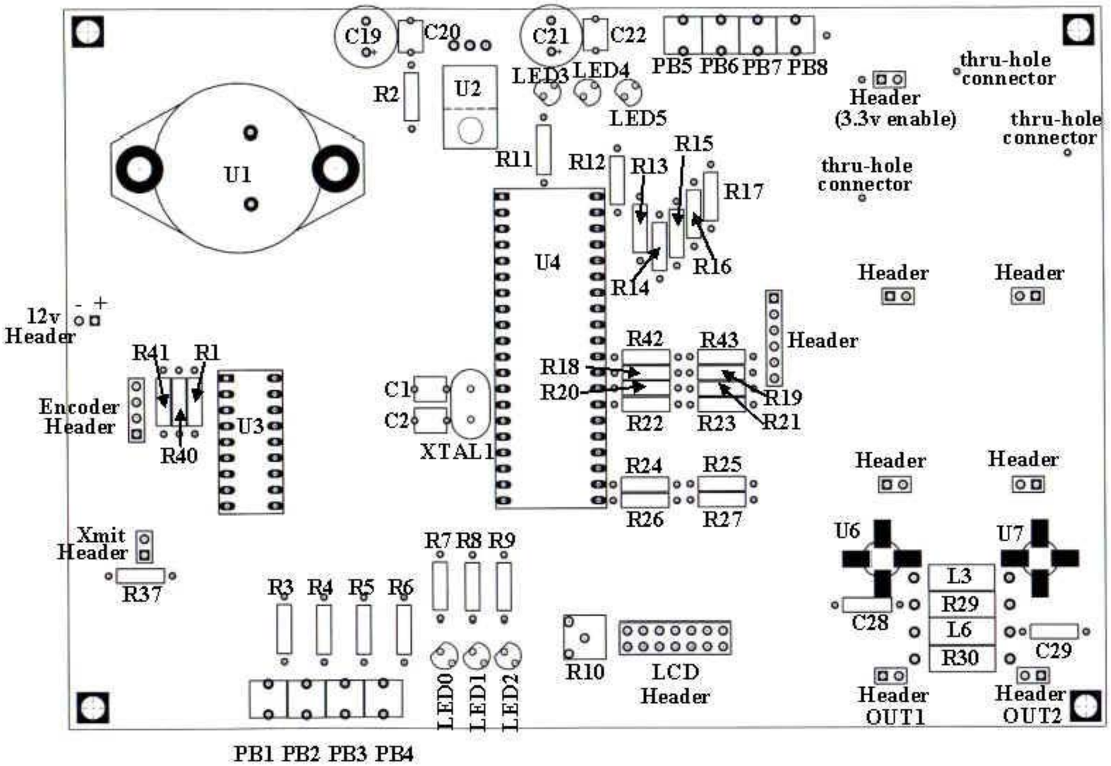
Bruce Stough, AA0ED

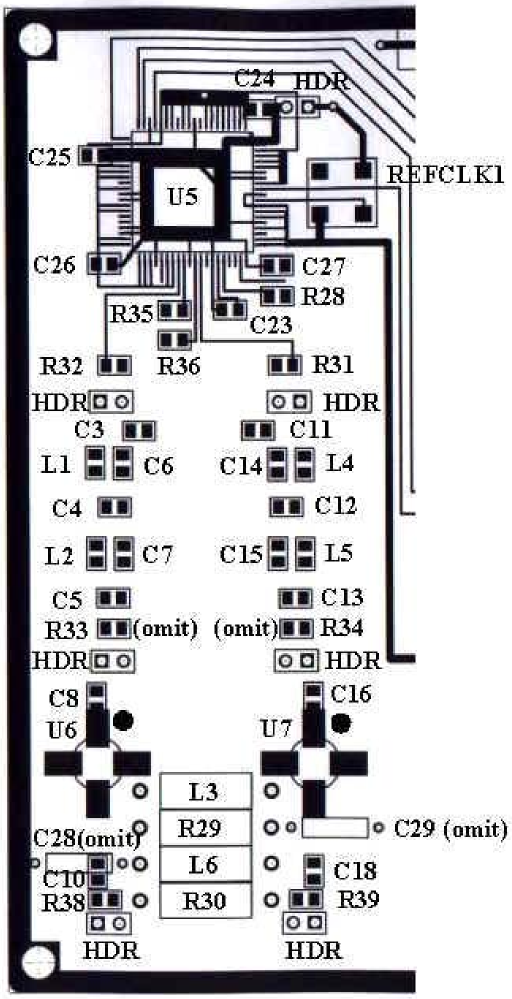
February 28, 2003

QTY	DESIGNATOR	PART	Part Number (DigiKey unless otherwise specified)
1	U1	5 volt regulator (TO3)	LM323K Steel-ND
1	U2	3.3 volt regulator TO220	LT1085CT-3.3-ND
1	REFCLK1	Crystal Oscillator 3.3v 125 MHz (SMT)	520-3953M-1250-BN (Mouser)
1	XTAL1	Crystal 20 MHz	CTX416-ND
6	LED0 thru 5	LED T1	P562-ND
8	PB1 thru 8	Pushbuttons 5.0mm SPST	P8083SCT-ND
6	R31, R32, R33, R34, R38, R39	Resistor, SMT, 0805, 51 ohm	311-51.0CCT-ND
2	R35, R36	Resistor, SMT, 0805, 24 ohm	311-24.0CCT-ND
1	R28	Resistor, SMT, 0805, 3.9k ohm	311-3.90KCCT-ND
2	R29, R30	Resistor, axial, 1/2W, 270 ohm	270H-ND
5	R18, R20, R22, R24, R26	Resistor, axial, 1/8W, 3.3k ohm	3.3KEBK-ND
5	R19, R21, R23, R25, R27	Resistor, axial, 1/8W, 5.6k ohm	5.6KEBK-ND
21	R1 thru R9, R11 thru R17, R37, R40-43	Resistor, axial, 1/8W, 10k ohm	10KEBK-ND
1	R10	Resistor, variable, trimpot, 1K ohm	36G13-ND
2	L2, L5	Inductor, SMT, 0805,	PCD1178CT-ND

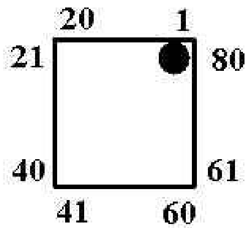
		0.33 uH	
2	L1, L4	Inductor, SMT, 0805, 0.39 uH	PCD1179CT-ND
2	L3, L6	Inductor, axial, 22 uH (shielded)	DN41223-ND
11	C8, C9, C10, C16, C17, C18, C23-C27	Capacitor, SMT, 0805, 0.1 uF	PCC1840CT-ND
4	C3, C5, C11, C13	Capacitor, SMT, 0805, 82 pF	PCC820CGCT-ND
2	C4, C12	Capacitor, SMT, 0805, 150 pf	PCC151CGCT-ND
2	C6, C14	Capacitor, SMT, 0805, 10 pf	PCC100CNCT-ND
2	C7, C15	Capacitor, SMT, 0805, 33 pf	PCC330CGCT-ND
2	C19, C21	Capacitor, axial, 10 uF tantalum	80-T356E106K016 (Mouser)
2	C1, C2	Capacitor, axial, 27 pF NPO	140-50N5-270J-TB (Mouser)
2	C20, C22	Capacitor, axial, 0.22 uF mono	BC1149CT-ND
2	U6, U7	ERA-3 MMIC, SMT pkg (by MiniCircuits)	(Dan's Small Parts) or (Down East Microwave)
0	U5	AD9854AST DDS	(Analog Devices)
1	U3	PIC16F628 4 MHz	PIC16F628-04/P-ND
1	U4	PIC16F877 20 MHz	PIC16F877-20/P-ND
1	Encoder	Optical Encoder (128 positions)	102-1007-ND (or alternate mechanical encoder: P10859-ND)
1	LCD	LCD (16 x 2 char, HD44780 controller)	628-L168200J (Mouser)
1		40-pin DIP socket (.600")	A411-ND
1		18-pin DIP socket (.300")	A403AE-ND

8		Headers (.100” spacing) (2-pins)	538-22-03-2021 (Mouser)
1		Headers (.100” spacing) (4-pins)	WM4002-ND
1		Headers (.100” spacing) (6-pins)	WM4004-ND
1		Headers (.100” spacing) (two rows of 7-pins)	538-10-88-1141 (Mouser)
1		Heat Sink (TO-3)	HS149-ND (changed to .75")
1		Heat Sink (TO-220)	345-1022-ND
1		PCB	

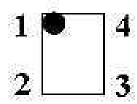

C19
C20
C21
C22
PB5 PB6 PB7 PB8
LED3 LED4
LED5
R2
U2
U1
thru-hole connector
Header (3.3v enable)
thru-hole connector
thru-hole connector
R11
R12
R13
R15
R17
R16
R14
U4
Header
Header
12v Header
- +
Header
R41 R1
R42
R43
R18
R20
R19
R21
R22
R23
Encoder Header
U3
C1
C2
XTAL1
R40
R24
R25
R26
R27
Header
Header
Xmit Header
R37
R7 R8 R9
R3 R4 R5 R6
U6
U7
L3
R29
L6
R30
C28
C29
R10
LCD Header
LED0 LED1 LED2
Header OUT1
Header OUT2
PB1 PB2 PB3 PB4



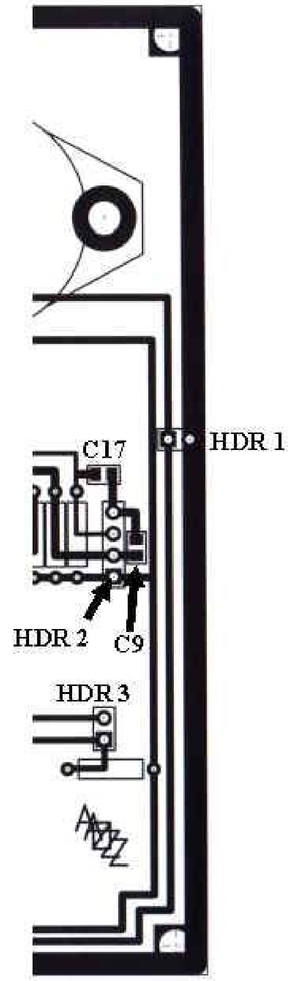
Surface Mount Parts Placement (Bottom side)

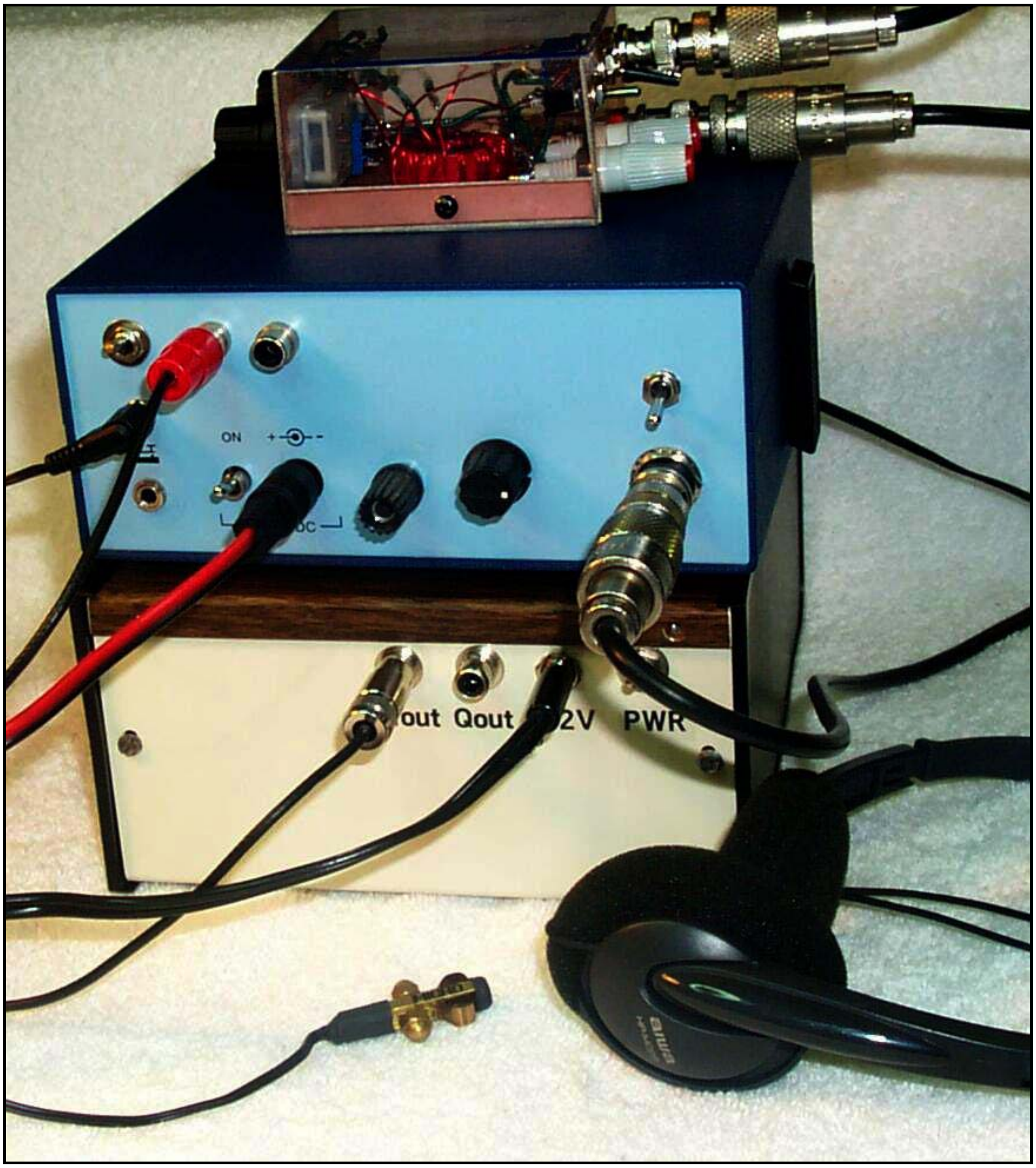


AD9854 Orientation

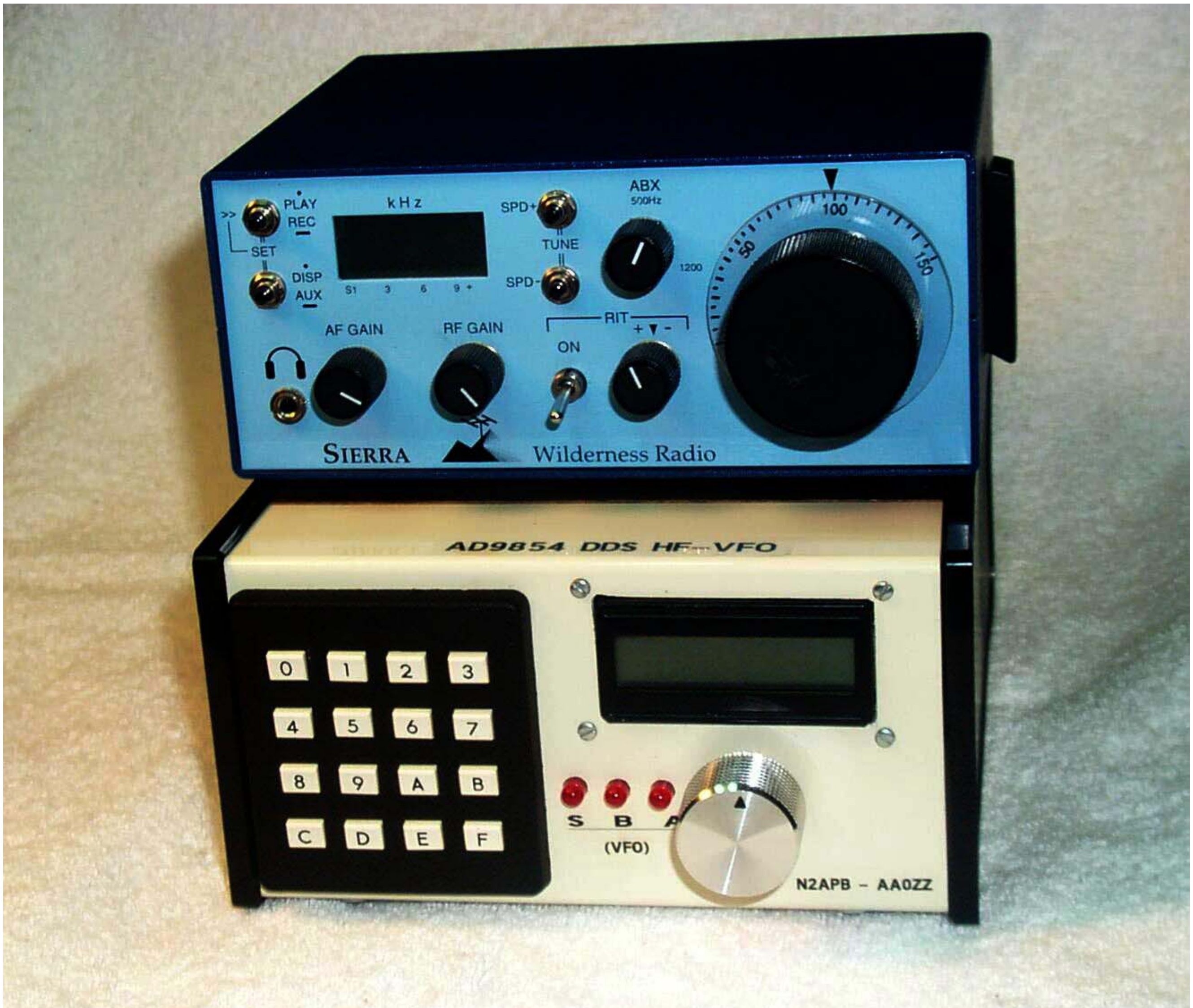


REFCLK1 Orientation





Here's the rear panel interconnections between the HF VFO and the Sierra QRP transceiver. This is a common operational setup used at the N2APB portable station with the NorCal BLT ATU on top, Sierra QRP Transceiver in the middle, and the HF VFO on the bottom. The headphones and small portable paddle are in the foreground, while the power supply (batteries) and antenna cable lead out of the photo. The Sierra was modified to accept an external VFO input that serves as the LO for the rig. (The small paddle shown in the foreground is a product made by KK5PY.)



Here's the front view of the stacked pair: the Wilderness Radio Sierra QRP transceiver (top) and the AA0ZZ 9850 High Performance HF VFO (bottom). During operation, frequency tuning is done (and displayed) on the external HF VFO, while the dial and display on the Sierra is not used.

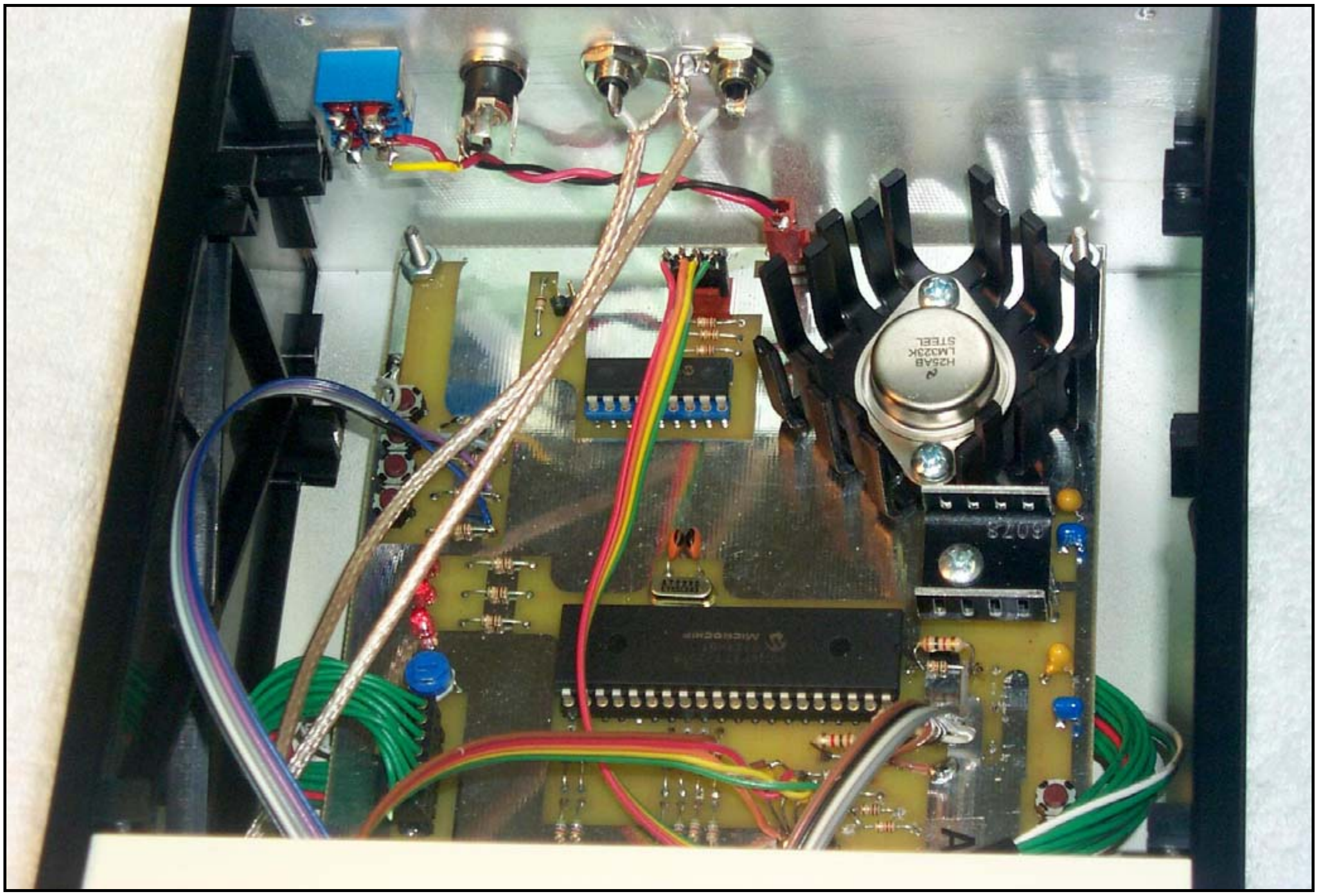
Building the IQ-VFO board into a fairly compact Ten-Tec enclosure makes for a great performing VFO and a great looking VFO on the bench! This model enclosure is no longer produced by Ten-Tec, but several others come pretty close (MW-7, MW-8 or TPB-49).

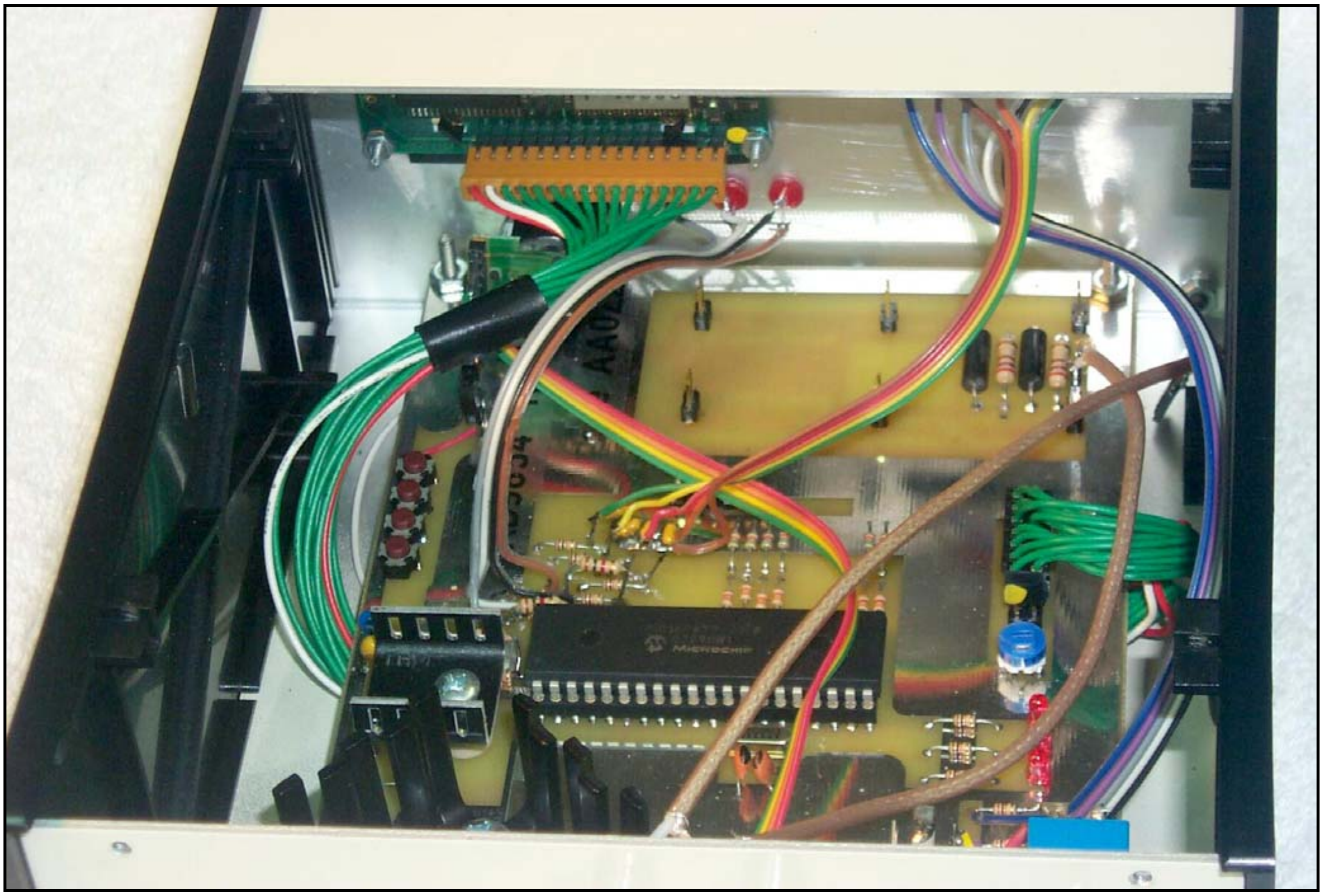
The hex keypad was installed by N2APB as a future feature for numeric input of frequency.

The rear panel provides connections for the two quadrature output signals (I and Q), the power jack and the power switch.


AD9854 DDS HF-VFO
12V

This inside view of the enclosure shows the HF VFO pc board to be a tight fit front-to-rear, but cable to the controls, display and connectors is easily accomplished above the board. The panel-mounted pushbuttons and LEDs are wired in parallel with their board-mounted counterparts.



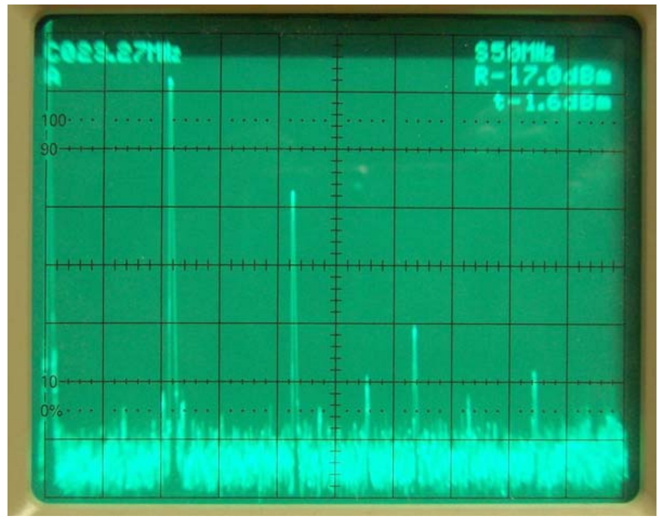


Craig Johnson, AA0ZZ may be reached by mail at 4745 Kent Street, Shoreview, Minnesota 55126, or by email at aa0zz@cbjohns.com

AmQRP.ORG HOMEBREWER

Craig Johnson, AA0ZZ
Jim Kortge, K8IQY

Harnessing Harmonics in the IQ-VFO



The spectral purity of a DDS-based VFO has been the subject of great debate in the technical community. Much of the discussion is related to earlier generations of DDS parts which feature Digital to Analog Converters (DACs) that are not as good as that on the AD9854 used in the currently popular IQ-VFO design. Designer AA0ZZ, along with noted QRP Hall of Fame member K8IQY give this topic some serious study.

Analysis of AD9854 output signal

Much of the speculation in the literature and the Internet that has been centered on the spurs that are generated as byproducts of the DDS devices. Most often the speculation proceeds to the assumption that a phase-locked loop (PLL) is necessary on the output to reduce the spurs.

The AD9854 has a 12-bit DAC while previous generation DDS's had fewer bits. For example, the AD9850 has a 10-bit DAC. Additional bits in the DAC are important in reducing the undesirable byproducts (spurs). Dynamic performance (Spurious Free Dynamic Range - SFDR) improves by 6 dB with each additional bit. (See Analog Devices Application Note AN-282.) The AD9854, with its 12-bit DAC, has an 80 dB SFDR.

This performance analysis was done for the purpose of determining the spectral purity of the DDS-based VFO as implemented in this IQ-VFO project.

Note that the IQ-VFO board includes 7-pole elliptical low pass filters between the AD9854 dual outputs and the MMICs, to suppress VHF emissions. The affect of these low pass filters is ignored in this analysis. The low pass filters' cutoff frequency is 30 MHz so it only affects (attenuates) signals that are higher than 30 MHz. Since we were trying to analyze the noise within the frequency spectrum that is lower than the LPF cutoff frequency we ignored those effects.

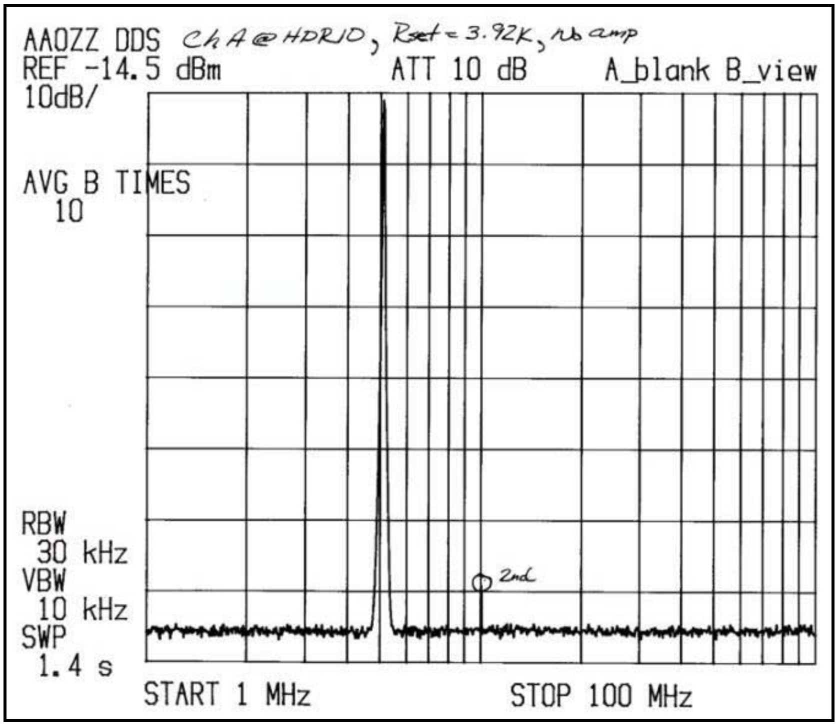


Figure 1

The output signal level of the AD9854 can be adjusted by changing the value of the resistor known as Rset. This resistor value is selected on the basis of the output signal level desired. The formula for selecting Rset:

Rset = 39.9 / Iout

The nominal value for Rset in the IQ-VFO is 3.9K, corresponding to an output current of 10 ma. This is in the center of the normal range of 8k (5 ma) to 2k (20 ma). Since the output load resistance is 50 ohms, 10 ma through 50 ohms corresponds to an output voltage of 500 mV RMS. Note that these output levels are not attainable when the load is not exactly 50 ohms. In this project, the load is not exactly 50 ohms when the low pass filter and MMIC are attached, so 500 mV is not observable at the junction of the AD9854 and the low pass filter.

Note that design decisions were made from the beginning of the project to maximize the AD9854 output spectrum purity. To this end, these design decisions were made:

- The AD9854's clock multiplier was not used.
  - The frequency of the DDS reference clock (125 MHz) was selected with the intent of keeping it well above the minimum (Nyquist) frequency. The Nyquist frequency is two times the desired fundamental frequency.
  - The output drive level was kept low.
  - All unused portions of the AD9854 were turned off and the inputs were grounded.

- Optional components that contribute to the SDFR, such as the DAC bypass capacitor and other bypass capacitors, were used.
- Careful isolation (single connection point) was done between the analog and digital grounds of the AD9854.

With these design decisions in place, what does the output of the AD9854 DDS look like? To answer this question, an Advantest R3361A spectrum analyzer was used to produce spectral plots to demonstrate how the DDS outputs change under varying conditions.

The first plot (Figure 1) shows the signal at Header 10, after the low pass filter, and no MMIC amp in the circuit. The only harmonic that is visible is the second harmonic and it is more than 65 dB lower than the fundamental. The other harmonics are down in the low noise level. In this case, the Rset value is 3.9k ohms. The signal level is 115 mV peak-to-peak, or 40 mV RMS.

The next plot, (Figure 2) shows what happens when the Rset value is changed to 15k, to reduce the AD9854 drive level. It is also taken at Header 10, after the low pass filter, with the MMIC removed. With these conditions, no noticeable harmonics or spurs can be seen; the output is very clean.

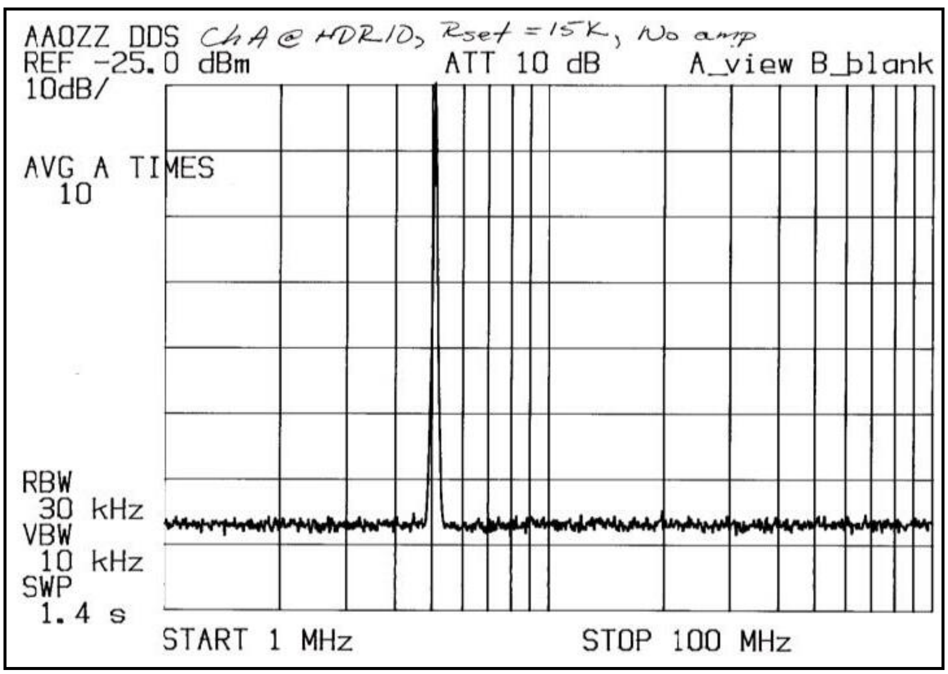


Figure 2

How about DDS harmonics/spurs? The SA output plots show they are all down 65 dB or better. This leads us to the conclusion that this IQ-VFO can be used as a sine wave frequency- generating source for an HF radio (transmitter and/or receiver in 1.8 to 30 MHz range) without additional filtering or PLLs to clean up the output signal. If is also evident that Rset has an effect on the generation of harmonics.

Lowering the DDS output level will lower harmonics and spurious signals.

What does the MMIC amplifier do to the output spectrum?

The next SA plot shows what happens when a MMIC amplifier is used with the AD9854.

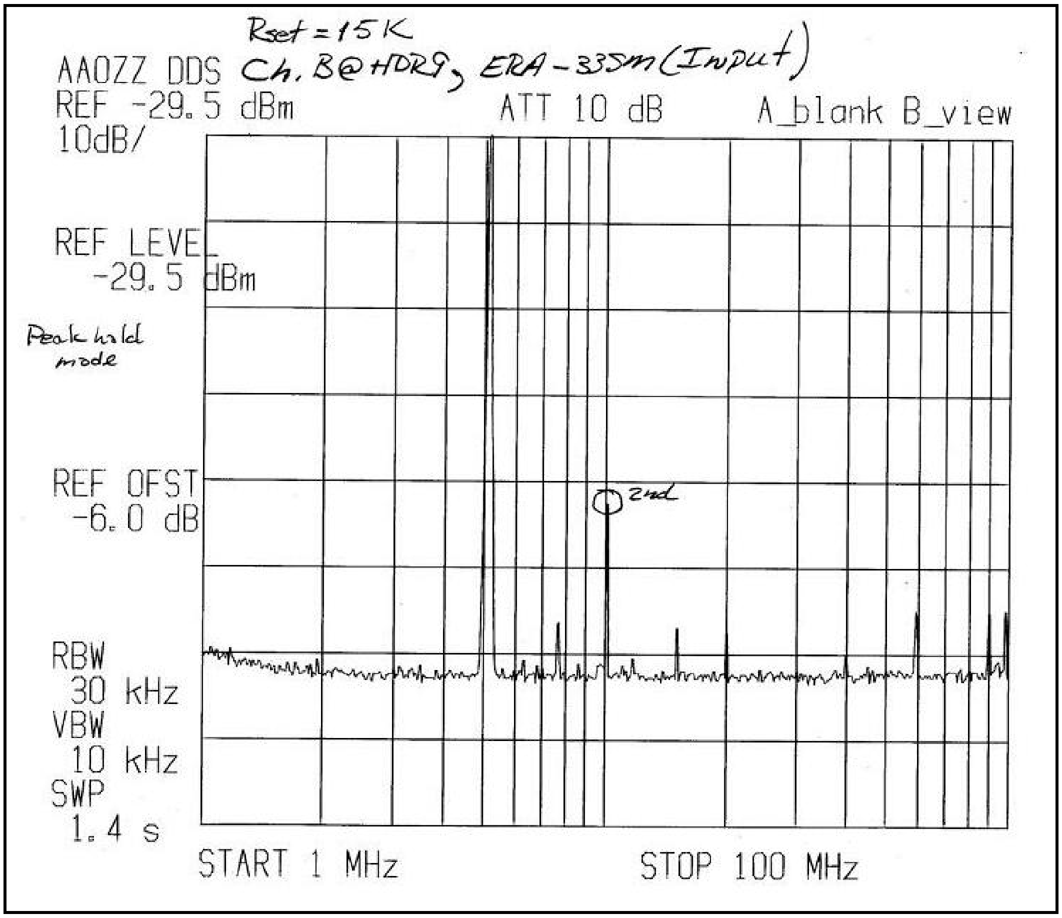


Figure 3

Figure 3 shows the output at Header 9, which is also the input to the MMIC amplifier. This is logically the same connection point as Header 10 but it is looking at the other output (channel) of the AD9854. Once again, it is after the low pass filter but this time a MMIC amplifier is connected. The second harmonic is now only 42 dB down, a degradation of about 25 dB, due to driving the input of the MMIC amplifier. Additional low level spurious outputs also appear. In this case, Rset is at 15k to reduce the drive and minimize these problems.

Figure 4 shows the signal at Header 11, the output of the MMIC amplifier. The second harmonic is now only 35 dB below the fundamental, and other byproducts are also seen. Rset is 15k in this case also. The signal level is now 2.4v p-p or 850 mV RMS (11.6 dBm).

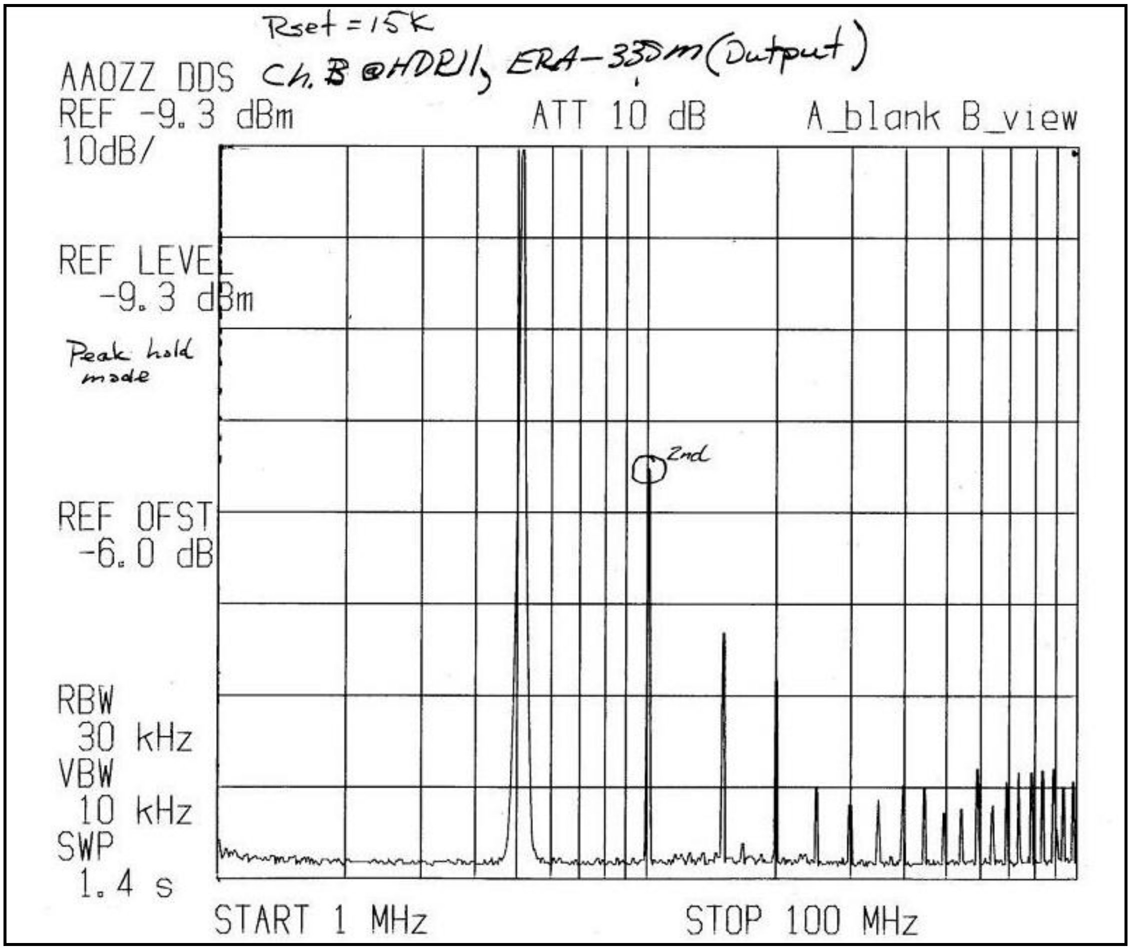


Figure 4

This leads us to the conclusion that MMIC amplifiers cannot be driven hard. If they are, the harmonic levels will be greatly increased.

Note that the MMIC used in these tests is an ESA-33SM. In other tests, the ERA-3 was shown to have similar characteristics.

Confirmation by Modeling the MMIC with ElectronicWorkBench

To investigate this phenomenon further, a MMIC was modeled with Electronics Workbench (EWB). Figure 5 shows the circuit that was used to represent the MMIC.

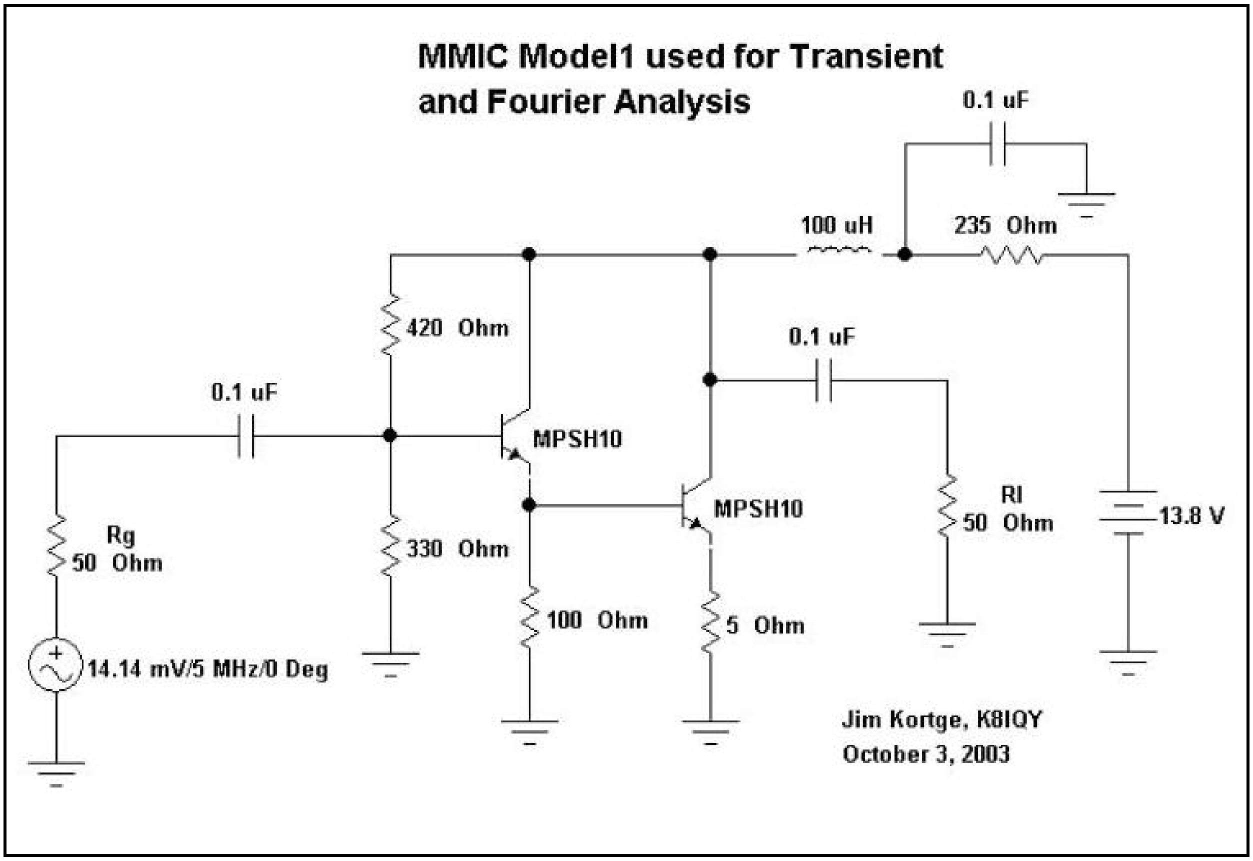


Figure 5

The generator in the simulation model is set to twice the desired input voltage since one-half of the power is consumed in the generator's internal impedance, Rg. The input of the MMIC model starts at the base of the first transistor and its bias resistors.

First, we analyze the MMIC when a input signal level of 7.07 mV RMS (20 mV p-p) is applied. Figure 6 shows the transient response of the model with this input level. The 20 mV p-p input level results in an output of 120 mV p-p. The modeling software can also provide spectrum analysis, showing the harmonics when this signal level is applied. The results, Figure 7, show a fundamental that has a magnitude of –25 dB, a second harmonic that is –82 dB (57 dB below the fundamental). The third and fourth harmonics are both about –102 dB, which is 77 dB below the fundamental.

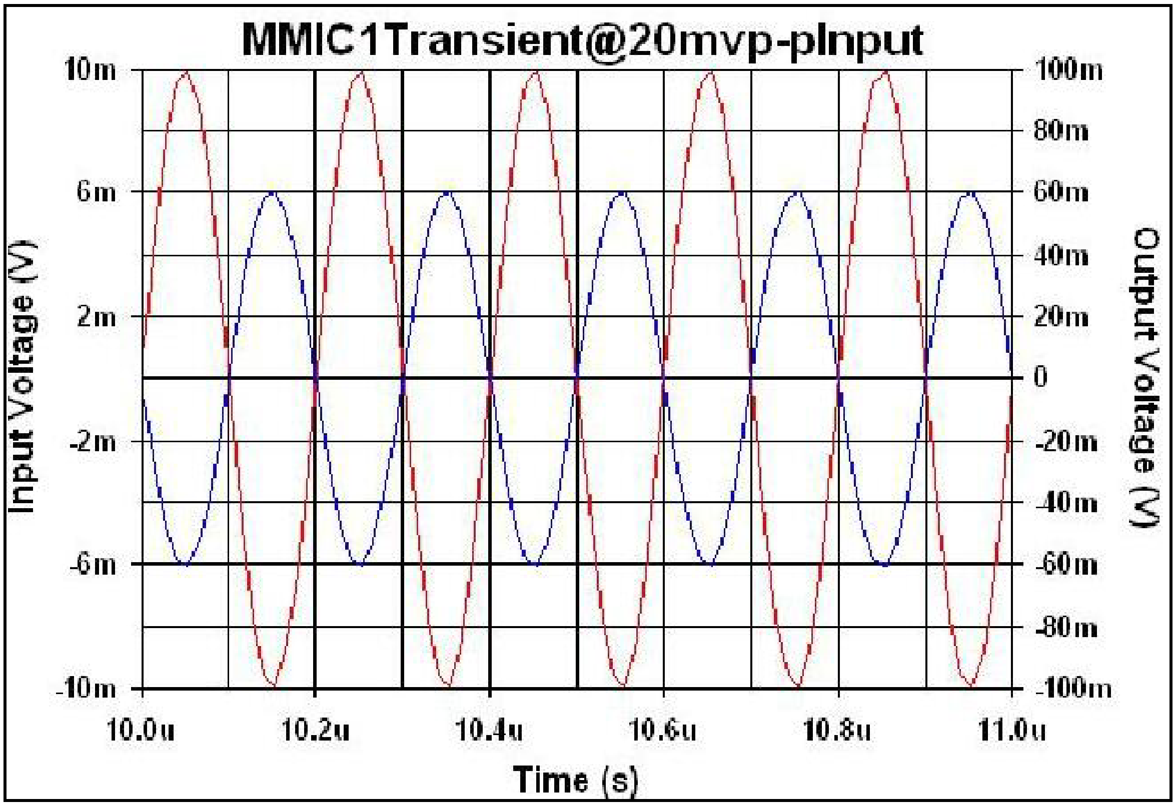


Figure 6

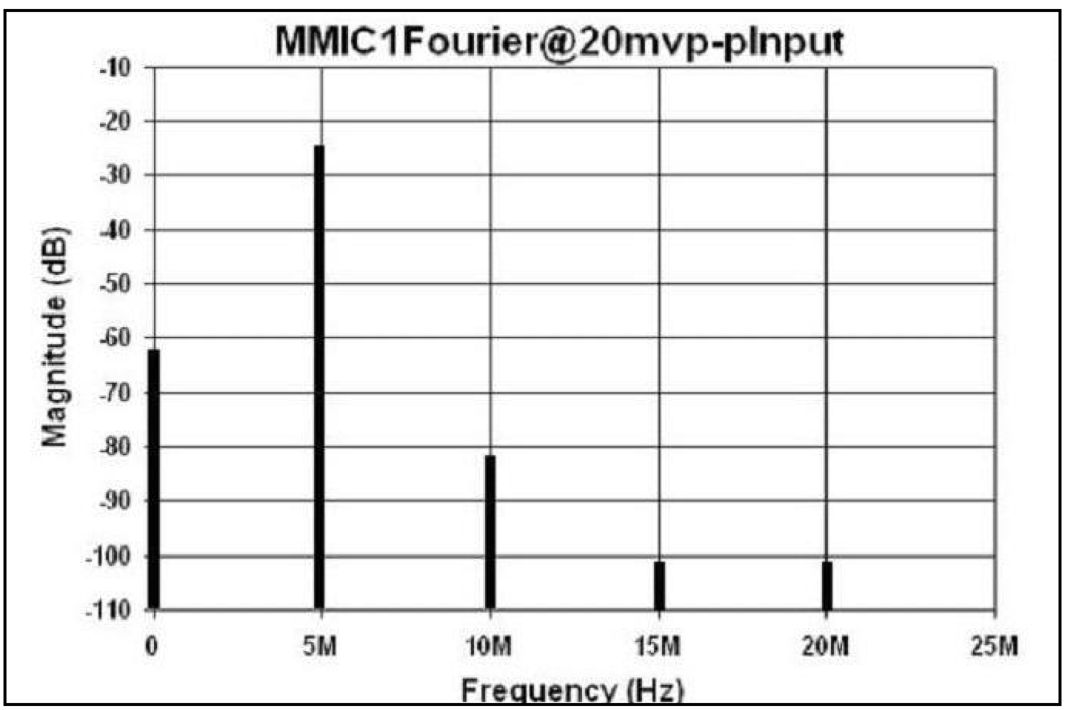


Figure 7

What happens when the MMIC is driven harder? This is answered in Figures 8 and 9. When the input is 200 mV p-p, the output is 1200 mV p-p. The spectrum analysis shows the fundamental magnitude to be –5 dBm and the second harmonic at –42 dBm, just 37 dBm below the fundamental. The third harmonic is at –61 dBm (56 dBm down from the fundamental) and the fourth harmonic is –88 dBm (83 dBm below the fundamental).

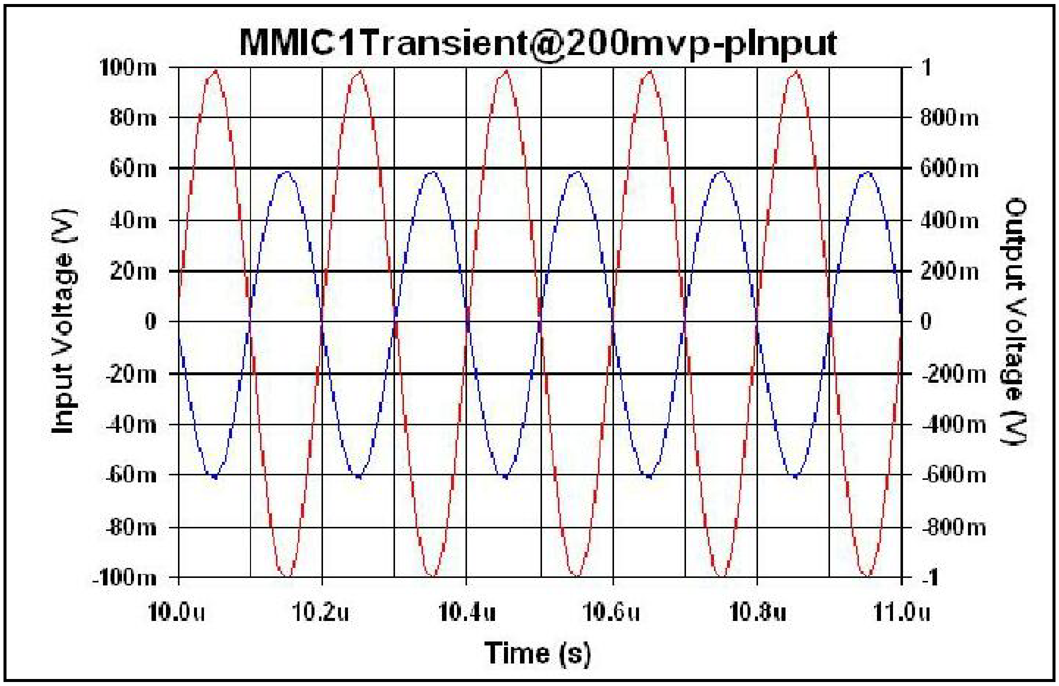


Figure 8

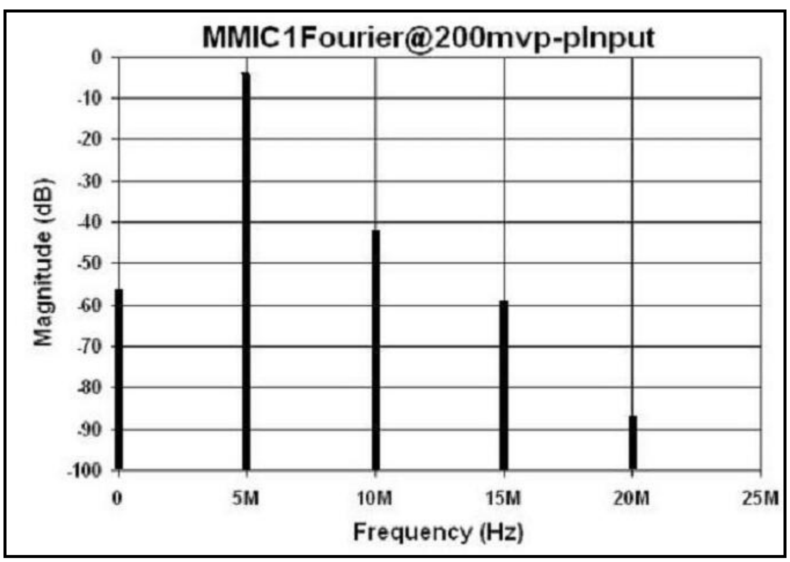


Figure 9

The harmonic levels are summarized in a chart (Figure 10) and a graph (Figure 11). The chart and graph also include values for the harmonics produced when the input drive level is doubled to 400 mV p-p. Now the second harmonic is only 28 dB below the fundamental and the third harmonic is only 38 dB down.

MMIC Model1 Performance Analysis

Input Voltage (P-P)	Output (dB)	2nd Harmonic (dB)	3rd Harmonic (dB)
0.02	-25	-82	-102
0.2	-5	-42	-61
0.4	1	-28	-38

Figure 10

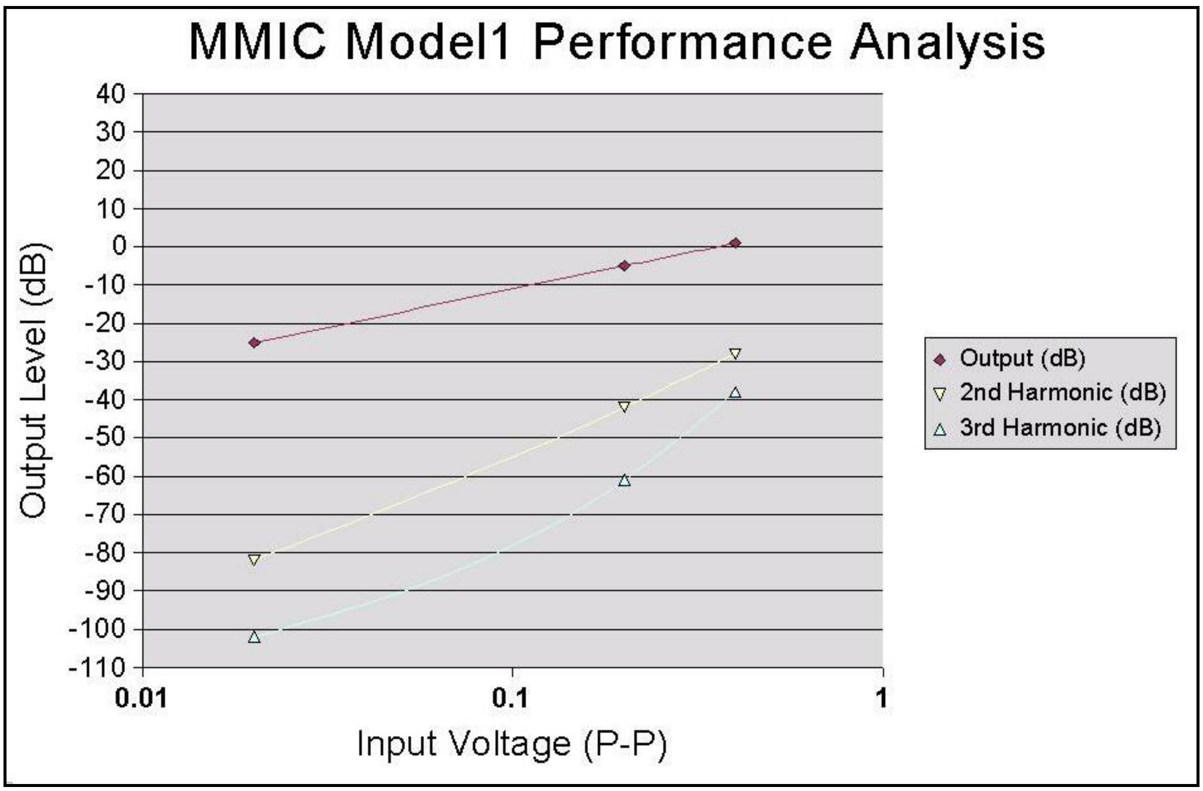


Figure 11

This EWB modeling confirms the previous observations that MMICs generate higher harmonics when driven with anything except very low levels.

Choices to be Made

What do these harmonic levels mean for an amateur radio VFO application? The builder has several options. The first would be to keep the drive levels very low and minimize the harmonics. However, these drive levels may be lower than what is required by most radio front ends.

Another option would be to replace the MMIC with a different type of amplifier having significantly better linearity. For example, a Norton lossless feedback amplifier or high performance op amp. Then the harmonics would again be be evaluated.

Still another option would be to drive the AD9854 at the higher level and use a low pass filter or band pass filter on the output to reduce the harmonics. Different filters would need to be selected for different frequency ranges.

In practice, filters will probably be required anyway!

Conclusions

1) The output of the AD9854 DDS is very clean. A Phase Lock Loop on the output is not necessary for HF radio applications.

2) MMIC amplifiers generate harmonics when driven at normal levels. These harmonics are still present but at much reduced levels when the MMICs are driven at a level that is much lower than rated levels.

3) When the IQ-VFO is used in a transmitter, band pass (or low pass) filters designed for the operating frequency range are needed to suppress harmonics.

Work to be Done

More experimentation is needed. These are some of the ideas that need testing:

- Replace the MMIC with a different type of amplifier having significantly better linearity. For example, a Norton lossless feedback amplifier or high performance op amp.
- Use band pass filters to further suppress spurious output. For multi-band operation, use relays to select different filters with appropriate frequency ranges.
- Try the IQ-VFO in different applications – with active and passive mixers, etc.

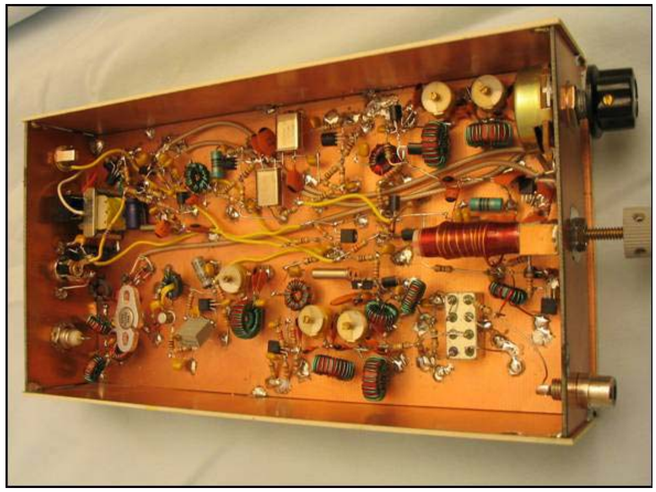
Craig Johnson, AA0ZZ may be reached by mail at 4745 Kent Street, Shoreview, Minnesota 55126, or by email at aa0zz@cbjohns.com

Jim Kortge, K8IQY may be reached by mail at PO Box 108, Fenton, Michigan 48430, or by email at jokortge@prodigy.net

AmQRP.ORG HOMEBREWER

Wayne McFee, NB6M

A Nuts and Bolts Approach to RF Design ... *Part 2*



This homebrew transceiver was the subject of a presentation made during the QRP Symposium at Pacificon 2003. I focused on the process of designing and building this transceiver from scratch by selecting portions of proven circuits and stringing them together in order to achieve the desired result. In the previous issue, I presented a step-by-step approach to designing and building this rig using some simple "nuts and bolts" techniques. This time I'll take you through the detailed circuit analysis, measurement and tuning procedures.

DESIGN OVERVIEW

When attention is paid to appropriate detail, building this or a similar transceiver is well within the ability of many homebrewers. Please refer to the schematic diagrams presented in Part I, as the discussion continues.

As can be seen in the photos, the entire transceiver was built "Ugly style" with discrete parts, using no Manhattan style pads or other devices.

Laying out the circuit in a logical and straightforward fashion, almost the same as the circuit diagram itself, kept interstage interference to a minimum, and allowed space for future modifications and improvements.

It has been shown that building RF circuitry onto a solid copper substrate helps to ensure that circuits perform more cleanly, spectrum wise. Evidence also shows that circuits constructed this way simply work better, and more as designed. An additional benefit of building this way is that the circuit can be built very rapidly, and then can be changed easily, as needed.

In this particular case, a simple superhet design was used as the base, and proven circuitry from many sources was added in order to complete the transceiver design.

Key ingredients for ensuring the success of the design were appropriate impedance matching and signal level adjustment between key stages. Taking appropriate steps to ensure the spectral purity of the transmitter output also provided for good results in the end.

An analog VFO using a permeability tuned variable inductor was chosen as the local oscillator (LO). This oscillator is quite stable, inexpensive, relatively easy to build, and allows for wide frequency coverage. In this case, the rig covers the bottom 165 kHz of the 40 Meter band.

VFO output was routed to the receiver mixer through an NP0 capacitor, the value selected so as to provide just over 5 Volts peak-to-peak to the mixer. Because the VFO Buffer Amplifier chosen for the transmitter chain required a signal taken from a source at DC ground potential, a 9 turn link was wound around the tuning coil, and the VFO signal was taken from there to the buffer amp.

Output of the buffer amplifier provides a little better than +7 dBm for the transmitter mixer chosen, a diode ring mixer, and also provides more than ample output for the FreqMite CW Frequency Annunciator. Although different mixers could be employed, the desire to use discrete devices and the spectral purity benefits of using a balanced mixer as a transmitter mixer, dictated the choice here.

The output from the Transmitter Oscillator was routed through a resistive attenuator pad so as to set the TXO injection level to the Transmitter Mixer to approximately -10 dBm.

One of the inputs to the diode ring mixer needs to be of sufficient amplitude to ensure that the diodes conduct fully on both halves of the input cycle. Hence, the +7 dBm level was chosen, as set by the VFO Buffer Amplifier. However, the second input needs to be kept quite low, about -10 dBm, so as to minimize spurious output from the mixer.

In addition to this, the mixer output was terminated in a 50-Ohm resistive 6 dB attenuator pad so as to provide a true 50-Ohm termination for all RF from the mixer. This was followed with a double tuned, 50-Ohm input to 50-Ohm output bandpass filter so as to select the proper signal from the mixer output for amplification by the transmitter amplifier chain.

A 50-Ohm input, to 50-Ohm output RF amplifier was added next in order to compensate for losses introduced by the 6 dB attenuator pad and the double-tuned bandpass filter, and to boost the signal to a usable level for the transmitter's driver amp. An LC impedance matching network follows this RF amplifier, in order to transform the 50-Ohm output of the RF amplifier to the 2.2 k Ohm input at the base of the transmitter driver amplifier.

Driver amplifier, power amplifier, and T/R circuitry, all borrowed from popular QRP transceiver designs, completed the transmitter. The initial transceiver design was completed with the addition of the muting circuitry.

Preliminary Results

Even in its first form, after initial construction, this transceiver showed very good promise. The receiver has a very low noise level and has plenty of sensitivity for the 40 Meter band.

Although the dual crystal IF filter might not be narrow enough for some, it does provide good opposite sideband rejection and has a nice, crisp, clean sound.

While the audio amplifier chain in this receiver does not supply a ton of audio, the audio output is quite sufficient and is very clean sounding as long as good quality stereo earphones are used.

The transmitter puts out 1.5 to 2.0 Watts, which is more than enough for many contacts. This lower power level was chosen because of readily available outboard power amplifiers for those times when band conditions warrant.

As with all construction projects, changes are made when the actual building starts. Some changes are made simply because of having substitute parts already on hand. Others are made because new ideas prompt improvements or additions to the original circuit. Further, some are made necessary because the final results may not be quite up to expectations.

In this project, once the injection levels to the receiver and transmitter mixers were properly set, and the receiver and transmitter frequency offsets were adjusted, the basic receiver and transmitter sections worked very well. The analog VFO is quite stable and it is really nice to be able to tune all of the CW portions of the band. Although it would certainly be nice to have a digital VFO, this one is so easy to build and gives such good results with so few parts that it is well worth using in more than one project. The addition of a FreqMite CW frequency annunciator, or another frequency reporting device, gives us the complete confidence of knowing exactly where we are in the band.

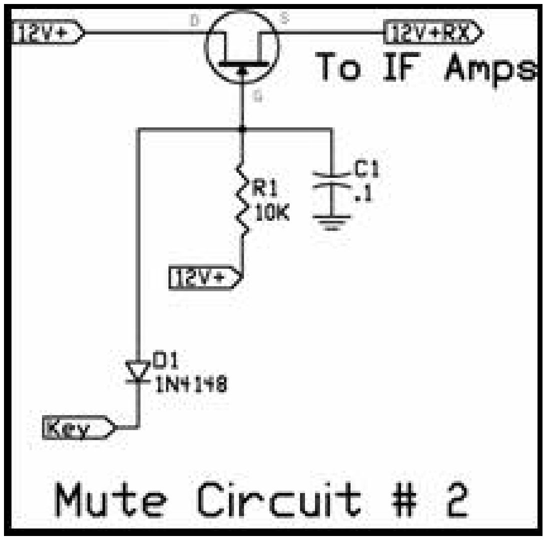
However, even without a digital readout or a FreqMite-type annunciator, it would be difficult to stray outside the band. The IF is exactly on 4.000 MHz, and when the LO is tuned to 3.000 MHz, effectively putting the rig on 7.000 MHz, a birdie is heard in the receiver, providing a very effective band edge marker. This birdie, plus the fact that the upper end of the tuning range is 7.165 MHz, helps to ensure that all transmissions stay inside the band.

One area of the transceiver that needed improvement was in the receiver muting circuit. A definite goal of this project was to be able to hear the transmitter's signal in the receiver. There are several advantages this, not the least of which is that you have a ready indicator that the transmitter is actually working. In addition, once the receiver and transmitter offset frequencies are adjusted so as to coincide, all one has to do in order to zero beat another station's frequency is to tune the rig so that the received tone matches the tone heard on transmit.

In this particular case, the JFET mute switch inserted between the audio preamp and the audio output amp did not quite produce the desired effect. Although it did provide some muting, the audio tone was too loud and had a strident quality, which indicated that muting was needed in previous stages of the receiver.

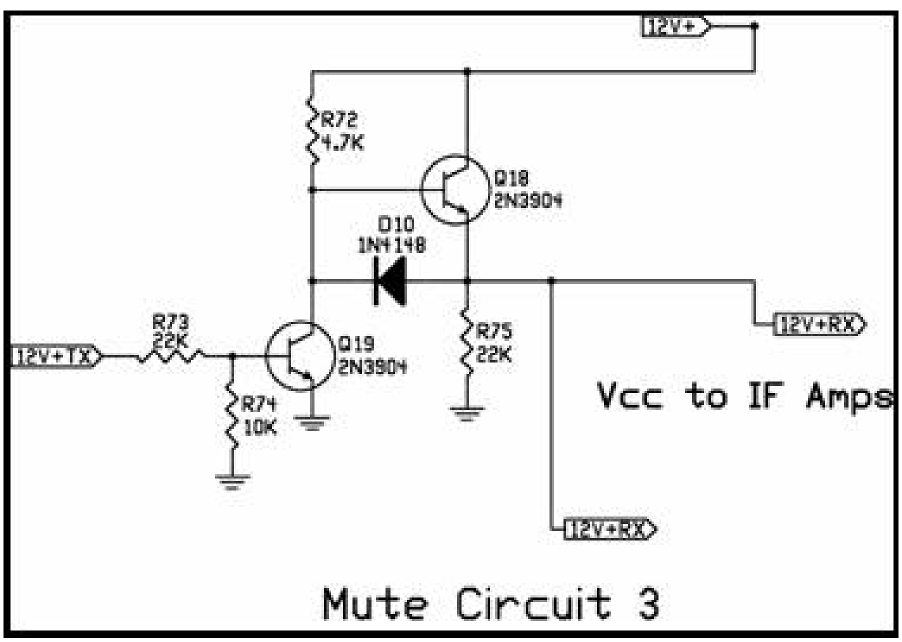
The first step in evaluating the problem and determining a correction was to move the muting circuit from between the two audio amplifiers, and place it between the product detector and the audio preamp. This did not correct the problem, so an experiment was conducted to see what audio level and quality would be had if Vcc was removed from the two IF amplifiers on transmit.

In order to answer this question, the 12-Volt leads to the two IF amps were unsoldered and disconnected. The results were that, on transmit, the audio tone quality was good and the audio level was just about right. So, the following circuit was inserted between the 12-Volt bus and the Vcc leads to the two IF amps.



The audio tone heard on transmit was louder than it was with the two DC leads simply disconnected. This was probably due to capacitive coupling, although this simple switching circuit did shut off the Vcc to the IF amps completely,

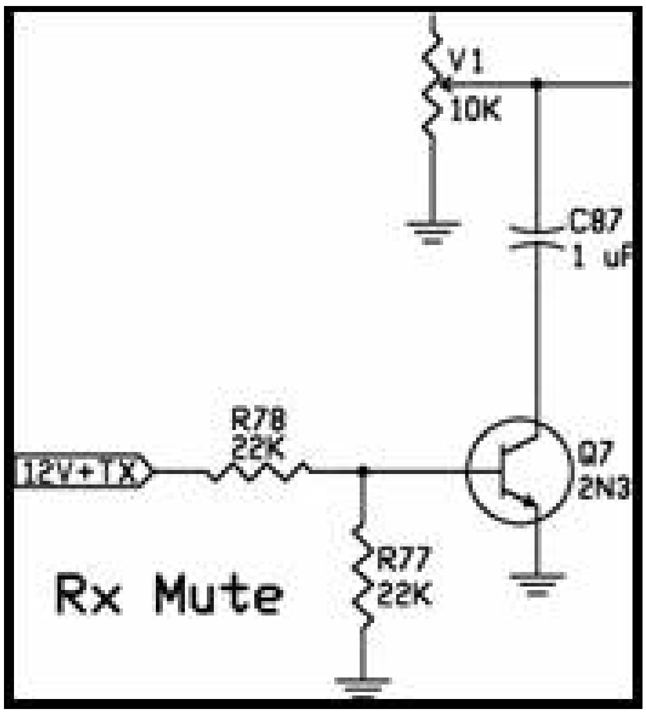
In an experimental effort to see if using another type of switching circuit would provide better results, the switching circuit shown below was installed. The results were the same. Although the audio tone heard was of good quality, it was still too loud at the highest settings of the receiver's volume control.



On-the-air experiments were conducted to determine if this was truly going to be a problem, or if it was a situation where the audio level on transmit was at an acceptable level at the normal settings of the volume control.

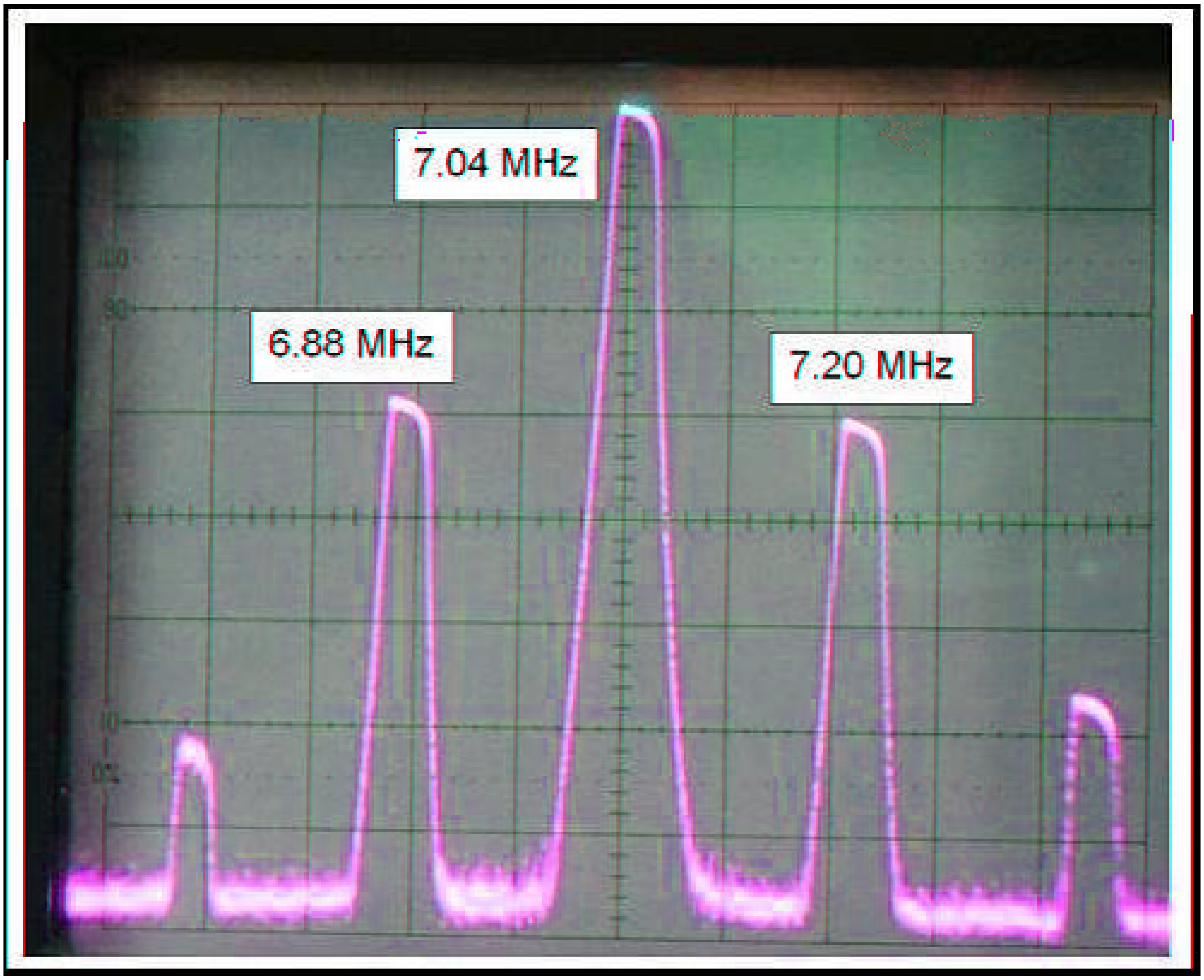
In practice, it was found that almost all of the time the volume control was at less than maximum, and the audio tone on transmit was at a comfortable level. However, in those situations where it had been necessary to have the volume control at maximum, one had to turn the volume down before transmitting, in order to keep the audio tone at a comfortable level.

Further experimentation showed that a keyed, bipolar transistor switch, connected in such a way as to short the audio from the wiper of the volume control to ground on transmit, gave a very satisfactory reduction in sidetone volume, and a much nicer sounding tone as well. Here is the circuit used.



Two other areas of concern cropped up, and showed the advantage of having a spectrum snalyzer on hand while the circuit was being built. Using the SA, I could actively check and modify stage performance as each stage was added.

First of all, the output of the transmitter did not initially meet FCC specs. There were two spurs – one at 6.88 MHz, and one at 7.20 MHz – that were -28 and -30 dBc (dB below the carrier) respectively, as shown in the spectrum analyzer photo below.

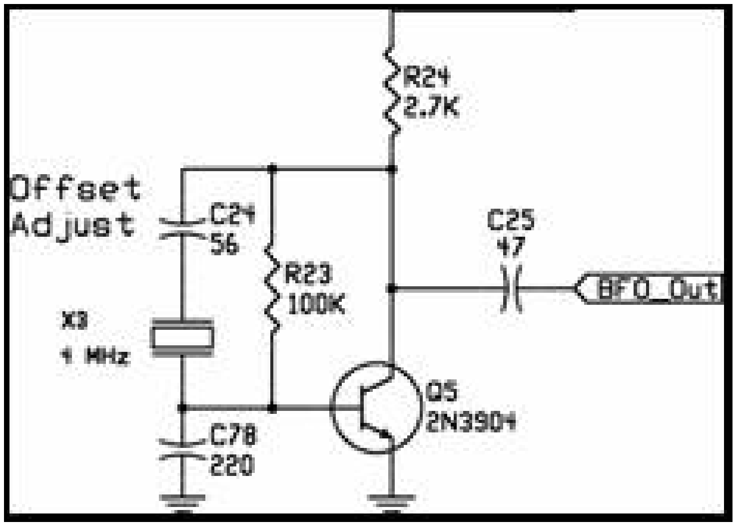


Spectrum analyzer shows spurs. (Test and photo courtesy of W7ZOI)

FCC specs for a transmitter with a power output less than 5 Watts require all spurious output to be more than 30 DB below the carrier, or main signal. The fact that the spurs were relatively close to the carrier frequency, definitely not harmonics of the carrier, and the fact that the output from the VFO Buffer had previously been checked, led to investigation of the output of the Transmitter Oscillator, which turned out to be the culprit.

Differences Between Receiver BFO and Transmitter Oscillator

The receiver's BFO frequency offset was accomplished with the addition of a 56 pF capacitor in series with the BFO crystal. This was done to utilize the lower sideband as the mode of reception in this 40 Meter rig.

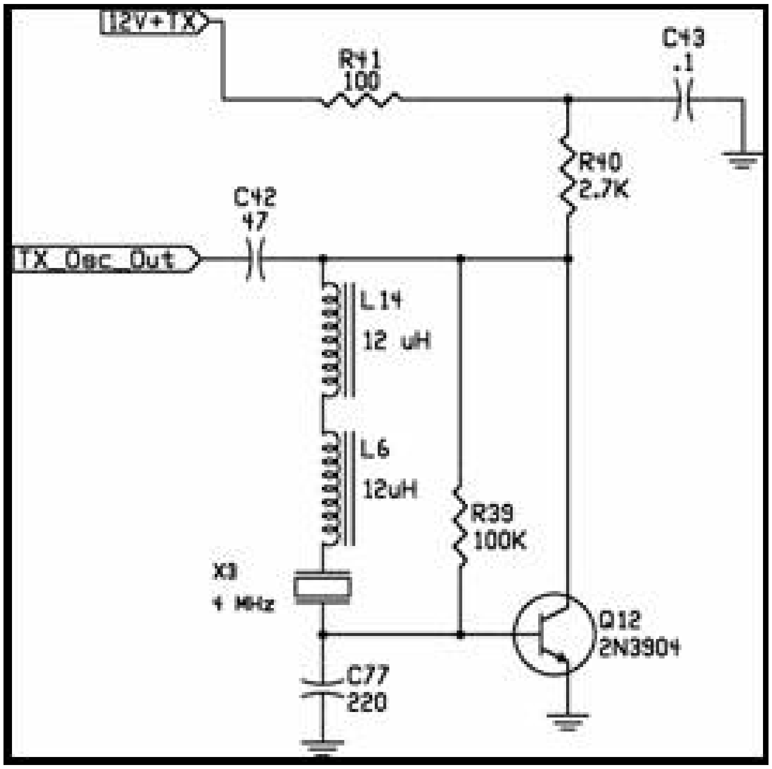


BFO

Adding capacitance in series with the crystal raised the oscillator frequency. The amount of capacitance was arrived at experimentally by adjusting the value so as to provide enough offset to give good rejection of the opposite sideband, while allowing for reception of the desired lower sideband.

The BFO output waveshape is a pretty nice looking sine wave, indicating little harmonic energy. This is borne out by the fact that the receiver's audio output is very clean sounding.

The Transmitter Oscillator's frequency was adjusted downward by the addition of two 12 uH inductors in series with the TX Oscillator crystal. This arrangement closely matched the transmitter's frequency offset to that of the receiver, but in the opposite direction frequency-wise. This places the transmitted signal very close to zero beat of a received signal when the rig is tuned so that the receiver's audio output tone matches the frequency of the transmitter's sidetone. This addition is shown below.

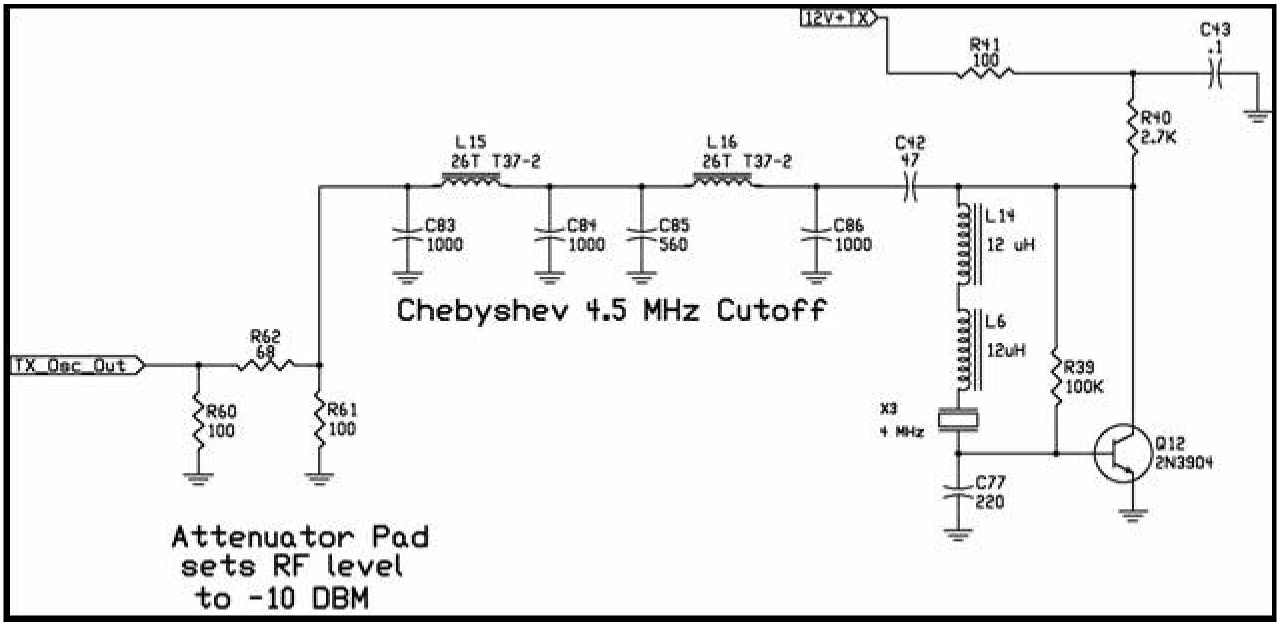


Transmitter Oscillator Mods

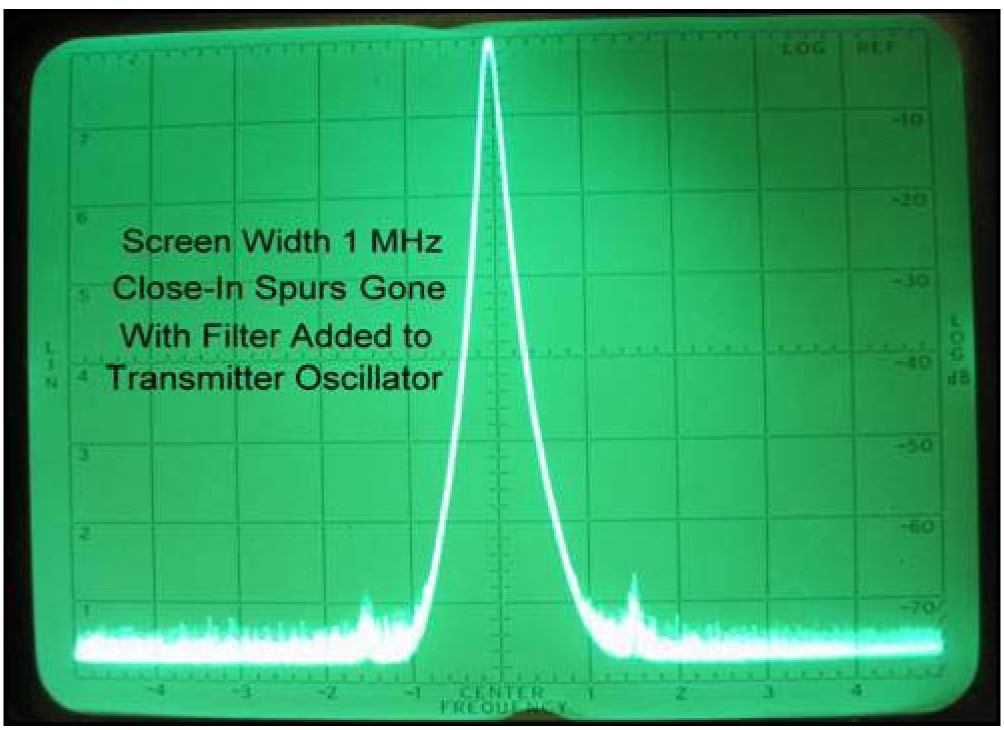
Again, the amount of inductance needed was determined experimentally.

In this case, on investigation, the waveshape of the Transmitter Oscillator's output was not a clean sine wave, indicating the presence of harmonic energy. As shown in the preceding photo of the transmitter's output on the Spectrum Analyzer, this harmonic energy caused strong spurs to appear relatively close to the main carrier frequency. The answer, and correction of the problem, was to filter the output of the Transmitter Oscillator, before its output was applied to the Transmitter Mixer.

A five-element Chebyshev filter, with a cutoff frequency of 4.5 MHz, was added to the output of the Transmitter Oscillator, as shown above.

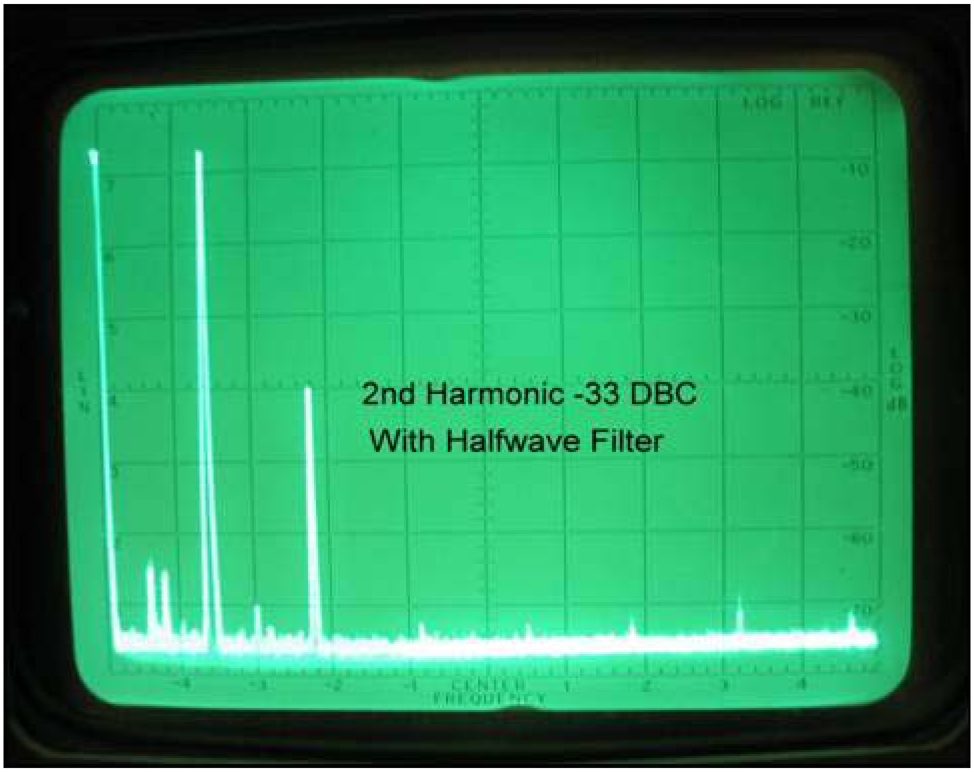


The addition of this filter completely eliminated the close-in spurs, as shown in the Spectrum Analyzer picture below.



(Test courtesy of WA6AYQ, photo by NB6M.)

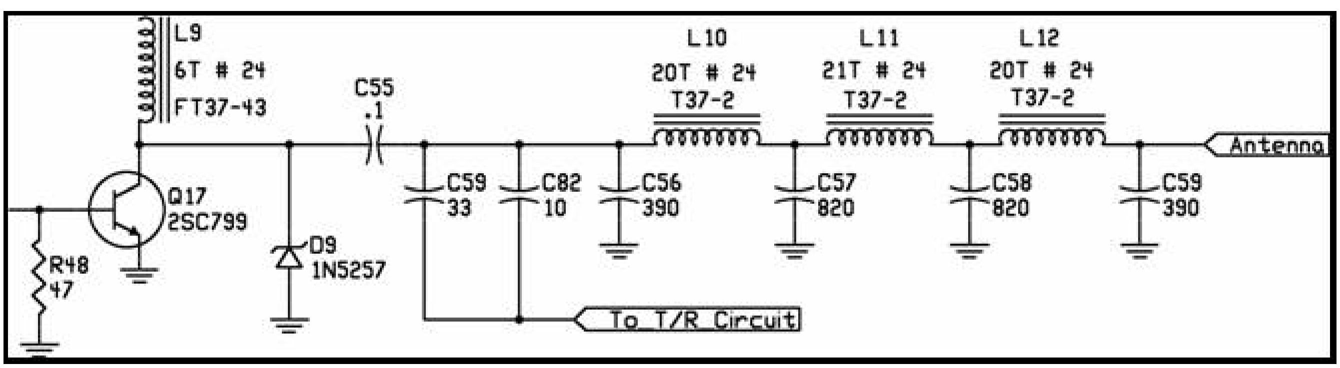
The second area of concern was that the wideband spectrum analysis of the transmitter's output showed that the second harmonic was at –33 DBC, in spite of the fact that all harmonics of the carrier were more than –30 dBc. Refer to the photo below.



(Test courtesy of WA6AYQ, photo by NB6M.)

While this was certainly within FCC specs for transmitters at this power level, further harmonic attenuation was desired.

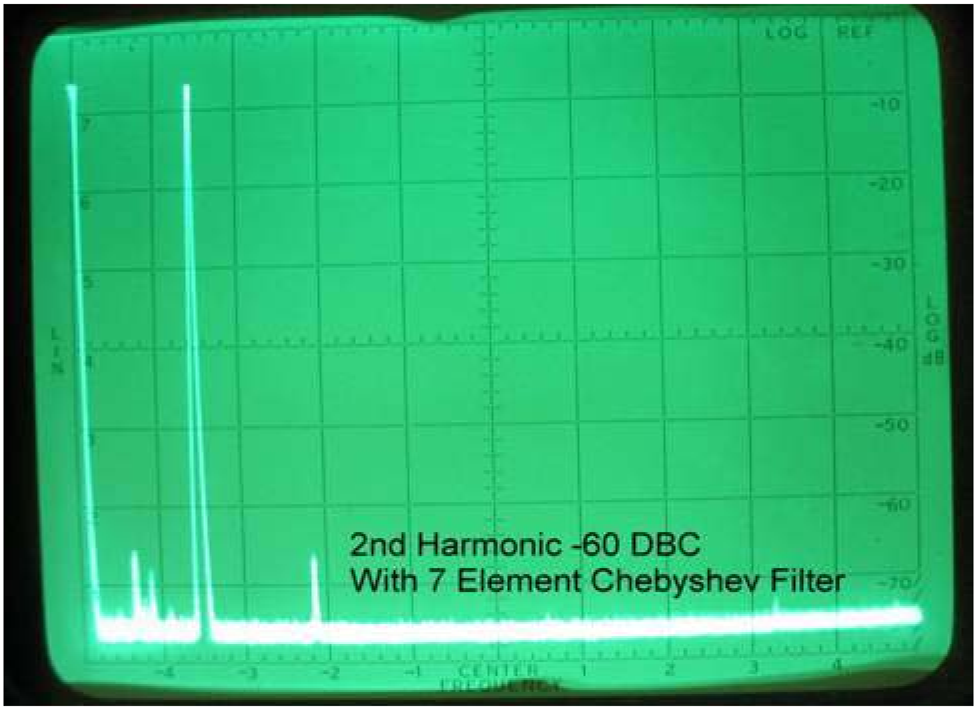
A five-element, halfwave output filter had been used in the transmitter's final amplifier stage. In order to further attenuate the second harmonic, and to see just how much difference it would make in the purity of the output, a seven-element Chebyshev filter was installed in the transmitter. This filter has a cutoff frequency of 8.5 MHz.



7-Element Filter

Although a seven element-filter is a bit of overkill in this case, it was interesting to see the improvement made in the spectral purity of the transmitter output.

As the Spectrum Analyzer photo shows below, the addition of this filter, all spurious output is now reduced to approximately –60 dBc.



(Test courtesy of WA6AYQ, photo by NB6M.)

Suggestions to Builders

Obviously, we all would love to have a complete set of laboratory test equipment such as spectrum analyzers, tracking Generators, and other high priced gear. Not all of us can afford to purchase such items, however, and we may not yet have the skills to build our own. What can one do with limited equipment in order to help ensure that a homebrew rig such as this operates as intended, and has a transmitter output that should meet current FCC specifications?

First of all, proper signal injection levels can be set for key stages such as the transmitter mixer. To do this the output of stages such as the VFO Buffer and the Transmitter Oscillator should initially be terminated with a 51-Ohm resistor, without running the signal further. Then the output can be measured with either a high impedance voltmeter and RF probe or with an oscilloscope, and the signal level adjusted as necessary.

The appropriate signal levels are often stated, such as the minimum of +7 dBm needed for one of the inputs to a diode ring mixer. These levels relate to the amount of power provided to a true 50-Ohm load, such as a 51-Ohm resistor used to provide termination for measurement purposes, and are not what would be read at the input of the mixer itself once the signal is properly routed.

In this case, the output of the VFO Buffer was adjusted by simply adjusting the number of turns of the link providing VFO energy to the amplifer, thus setting the output at just approximately +8 dBm.

In the case of a homebrew diode ring mixer using 1N914 devices, it takes a minimum of 0.7 Volts to make one conduct, which means that one of the two signals fed to the mixer should be at least a little over 1.4 Volts p-p. 5 milliwatts, or 0.5 Volts rms, is 1.414 Volts p-p, which is 6.9897 dBm. Therefore, it would seem reasonable that in this case, the VFO Buffer output should be set just little above that level. 7.2 milliwatts, or 0.6 Volts rms, is 8.5733 dBm, and should be adequate for the 1N914 diodes used in a homebrew mixer. Remember, this measurement is made when the output is connected across a 51-Ohm resistor to ground, not when it is connected to the mixer.

This is also true of setting the injection level from the Transmitter Oscillator. A 51-Ohm resistor to ground is connected to the output and the measurement is taken. The dBm level is calculated, and a 50-Ohm resistive attenuator pad is selected to provide the needed reduction. The pad is inserted between the Oscillator output and the mixer. Resistor values for pi-type 50-Ohm attenuators are easily found in such references as the ARRL Electronics Data Book.

As an example, let us say that we measured 0.45 Volts rms across the 51-Ohm resistor terminating the oscillator output. The 0.45 Volts rms signal is 4.05 milliwatts (rms, squared, divided by 50), which is 6.07 dBm (4.05, divided by 1, LOG, times 10). Let’s round that off to +6 dBm.

In order to set the injection level to –10 dBm, we need to reduce the signal level by 16 dB. According to the table on page 5-2 of the ARRL Electronics Data Book, we need resistances of 153.8, 68.8 and 153.8 Ohms, respectively, for a 16 dB, 50-Ohm impedance, pi-type attenuator. We use the closest values, 68 Ohms and 150 Ohms, and there we are.

Now, if one doesn’t have a Spectrum Analyzer on hand, in order to evaluate the outputs of stages such as the VFO Buffer and Transmitter Oscillator, how can we tell if the signal coming from these stages is relatively clean?

An Oscilloscope is needed in order to check the wave shape of the signals when they are terminated in the purely resistive 50-Ohm load. If the wave shapes appear to be clean sine waves, then there should be little harmonic energy. If the wave shape is distorted, then harmonic energy is definitely present, and a filter such as what was used in this rig between the Transmitter Oscillator and the resistive attenuator, should be installed. A 51-Ohm resistor should be connected from the output of the filter to ground and the waveshape should be checked again before routing the signal to the mixer.

The mixer itself will distort the wave shape of the inputs once they are connected, and for that reason, checking input wave shapes at the mixer will be misleading.

The use of the resistive 50-Ohm impedance, 6 dB attenuator at the output of the transmitter mixer helped ensure that harmonics of the two signal inputs were properly terminated. The double-tuned bandpass filter following the attenuator also selected the proper signal for amplification to the transmitter output.

The transmitter output filters with higher harmonic rejection can certainly help ensure that the signal radiated out the antenna is as clean as possible. Thus using the 7-element Chebyshev filter instead of a 5-element half wave filter improved the signal quality.

Thoughts for the Future

While this homebrew transceiver is fun and easy to operate, I am quite sure that it has not reached its final state of development.

Different audio amplifier circuits could be added to give a speaker output with plenty of volume. RIT could also be added to the receiver. The keyer should be built in. The FreqMite or another type of frequency annunciator would also be easy to build-in. Different types of mixer and amplifier circuits can be tried. Of course, there are other bands to build for, and on which to enjoy the operation of a totally homebrew rig.

This little rig has provided not only a wonderfully informative learning experience, but now provides the satisfaction of being able to say "rig HB, from scratch" to each person in QSO. There is tremendous pride in knowing that the rig meets and surpasses expectations on almost all levels. Even more satisfying is the joy of sharing information and ideas with others who may be interested in trying scratch building as well.

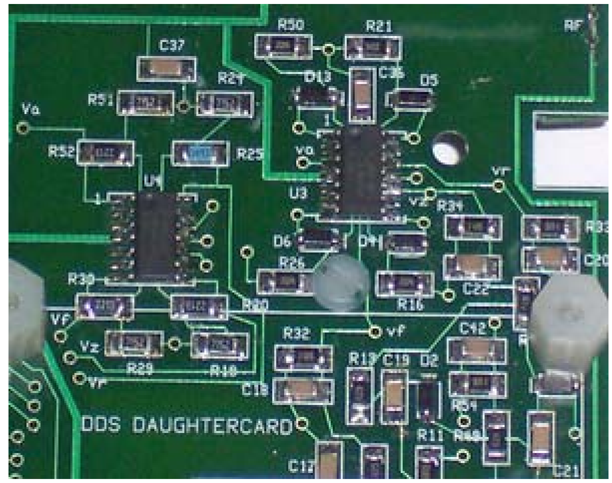
Wayne McFee, NB6M may be reached by mail at 2379 Saint George Drive, Concord, California 94520, or by email at NB6M@aol.com

AmQRP.ORG HOMEBREWER

Joe Everhart, N2CX

RF Power Meter Cookbook ... Part 3

This installment outlines test methods and simple circuits used to test RF power meters when you don't have access to expensive lab-grade equipment. N2CX focuses on testing below 30 MHz to demonstrate principles without resorting to special techniques.



Implementation of the N2CX power meter on a corner of the Micro908 pc board.

When this series of articles began it seemed that a couple of installments would be sufficient to cover the subject. However in the course of writing it became obvious that the subject matter was large enough that more than two were needed to fully discuss the principles and applications. The last segment stated that this time around we would get to the final design. In working out the details of that design, a large amount of time was needed to do the preliminary testing required to check out the power meter performance to be sure that it was as expected.

This installment will outline some test methods and simple circuits that a homebrewer can use to test RF power meters without the use of expensive or tough to find lab-grade test equipment. The focus will be on testing below 30 MHz to demonstrate the principles without the need to employ special techniques. The same ideas can be used well into VHF by minimizing the length of interconnecting leads and optimizing circuit layout.

When designing and building test gear homebrewers need to make maximum use of test equipment and components they have or can get easily. Best success comes when the basic instruments and parts have known performance that can be extended to other uses.

That sounds kind of abstract, so here are some examples to start with. We can buy reasonably good DMMs (Digital MultiMeters) for well under $100. They offer DC and AC voltage and resistance measurement accuracies on the order of 1%. In addition, inexpensive resistors and potentiometers can be used well into the HF part of the spectrum with good accuracy. Ordinary diode detectors can be calibrated using DMMs and some special techniques to accurately measure AC and RF voltages.

Using the above elements we will make some DC and low frequency AC measurements that can be done directly with the DMM to calibrate our test setup then use the calibrated test setup for RF measurements. Several test methods will be described in the following sections. They are complete in themselves though they may not apply to every situation. Sufficient description of the underlying principles is given so that the reader can adapt the methods shown to his own needs.

Diode Detector Calibration and Testing

The half-wave diode detector we discussed back in the second article in the series is a real workhorse. It allows us to easily measure RF by converting RF sine waves to DC. The DC output can then be read very accurately with a DMM. It suffers however from inaccuracy at low input levels due to inherent diode nonlinearity. However, for inputs of 1V rms or higher, it can be calibrated by measuring its output at various levels and making a calibration chart. The calibration chart is then used afterward to correct readings made using it to compensate for the non-linearity. That may sound like a chicken and egg situation, but it really is not. The key to making up the calibration chart is to perform the measurements at low AC frequencies where we can easily generate and measure the levels needed.

Figure 1 shows the test circuit used.

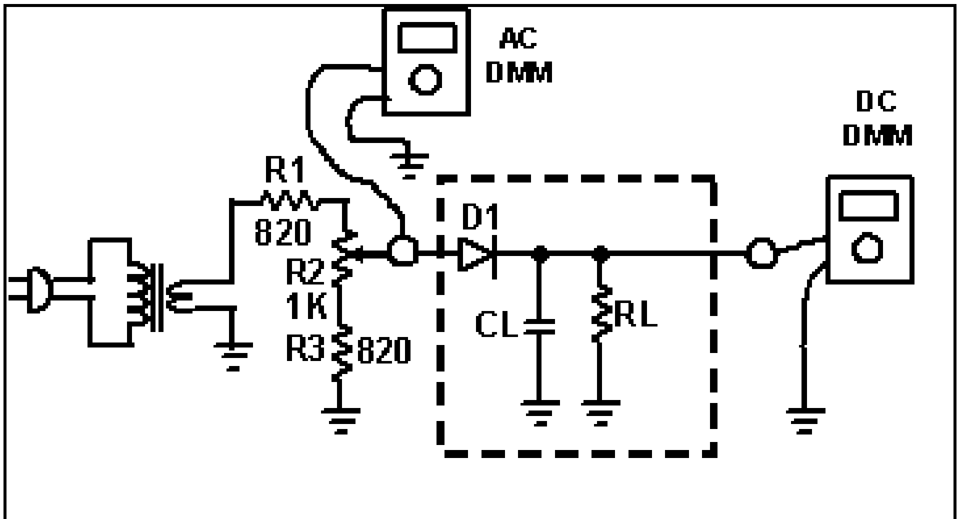


Figure 1 - Diode detector tester

The AC input is provided by a 24 volt filament transformer if you can find one or by two 12 volt transformers with their secondaries connected in series. The transformer supplies this AC voltage to a resistive network and potentiometer that give the test circuit an adjustment range of 0 to 24 V AC. The adjustment range is segmented due to the power rating of the potentiometer. Common potentiometers have a maximum power rating of only ¼ to ½ watt so the voltage across the pot has to be limited to no more than about 15.8V. The entire range can be achieved by shorting out one or the other resistors to keep the potentiometer from burning up. Either one or two meters are required. One of them is used to measure the AC voltage out of the potentiometer and the other is used to measure DC voltage from the diode detector being tested. The one measuring the AC voltage is used as test equipment while the DC meter is whatever will be used with the diode detector in its final application.

The detector being tested is shown in the dotted block in Figure 1. It is simply a diode, a filter capacitor and a load resistor. The filter capacitor is charged to the peak value of the input sine wave minus the diode drop and the load resistor allows the circuit to adjust itself to that peak value as the input voltage is adjusted. The detector diode can be either a germanium 1N34 or a Schottky diode such as a 1N5711. Schottky diodes have a somewhat higher forward drop as compared to that of Germanium diodes but they have more repeatable characteristics and will allow use of much higher load resistors.

The value of the filter capacitor depends on the load resistor used. It must filter RF from the detector DC output. As a conservative figure the RC time constant (R_L * C_L) should be 100 times the period of the input sine wave. For example, at 60 Hz, with an Rl of 1 megohm, we want RC to be 100 * 1/60Hz so C_L needs to be at least 1.67 µf. As a practical matter the capacitor must have a very high leakage resistance so it must have either ceramic or plastic film dielectric. Ceramic capacitors this high in value are uncommon so a good choice is either a single 2.2 µf or two 1 µf Mylar capacitors in parallel. Likely the detector will already have an RF bypass cap such as a .01 µf or larger ceramic filter capacitor so simply connect the larger capacitor across the output terminal during testing then disconnect it when testing is completed.

A short step-by step procedure to test the detector

1. Begin with no AC applied to the test circuit and connect the detector to the center arm of the potentiometer. Adjust the potentiometer so that the center arm is at the low end of its range.

2. Connect the DMM to the center arm of the potentiometer and set it read AC volts on the next scale above 25 V rms.

3. Connect a jumper across resistor R1 in the test circuit. This will limit the output voltage to about 12 V or so.

4. Apply AC to the circuit and adjust the potentiometer so that the DMM reads exactly 1V RMS.

5. Set the DMM range switch as appropriate for its most accurate reading.

6. Read and record the DC indication on the detector output meter.

7. Set the potentiometer so that the DMM reads 2V rms then read and record the DC output indication.

8. Repeat Step 6 at each one volt interval up to 12V AC. Reset the DMM range switch as necessary.

9. Remove the AC input voltage. Set the potentiometer to its lowest setting.

10. Remove the jumper across R2.

11. Reapply the AC input voltage.

12. Set the potentiometer for a 13VAC reading on the DMM then read and record the output meter DC reading.

13. Repeat step 11 at one volt increments up to 24V AC.

14. Remove AC input from the circuit.

At this point calibration readings have been made at one volt increments from 1 VAC to 24 V AC. The readings at the lower voltages may show some nonlinearity and should become more linear at higher steps. The detector has now been characterized at 60 Hz and should retain its calibration up through 30 MHz if short leads and a clean layout have been made.

Compensated Detector Test and Calibration

The compensated detector described in earlier installments of this series uses and operational amplifier with diode feedback to overcome detector non-linearity at levels down below 100 mV input. It s behavior above about 1V input is predictable so testing can be limited to lower levels.

This requires a more sophisticated method. While the testing could be performed at 60 Hz, using a variation of the method above this requires multiple large value capacitors and the DMM can become inaccurate at low AC voltages. The answer is to do testing at a frequency in either the audio range or low RF range. If a suitable RF source with an adjustable 1V output is available, by all means use it with the circuit to be described shortly. If not, testing can be done at audio frequencies and performance at RF should be adequate.

A suitable RF source is a good quality RF signal generator (a DDS signal generator is ideal) or a low power transmitter. Either one needs to produce a signal of at least 1V rms with harmonics and spurious signals suppressed by at least 30 dB. The exact frequency is not critical though using 3.5 MHz lessens errors due to lead lengths and stray capacitance in the test setup. Make sure that either source sees its proper load impedance to preserve output signal purity. The test setup is rather high impedance so it should add minimal loading. If a low power transmitter is used the level delivered to the test circuit should be kept to no more than 3 or 4V rms. To gauge this, consider that a 1 Watt transmitter produces an output level of 7.07 V rms. So a 6 dB or greater attenuator will be needed to give the desired input to the test circuit.

Testing at audio frequencies can be done without too much difficulty though a high-purity sine wave audio signal source is needed that will produce at least 1 V rms into a 1000 ohm load. If such a signal generator is not available, one option is to use a PC running a program such as Digipan (Ref 1) to produce the audio sine wave output. Since the computer's sound card will likely not have a high enough undistorted output level, a power amplifier such as the one in Figure 2 will be needed.

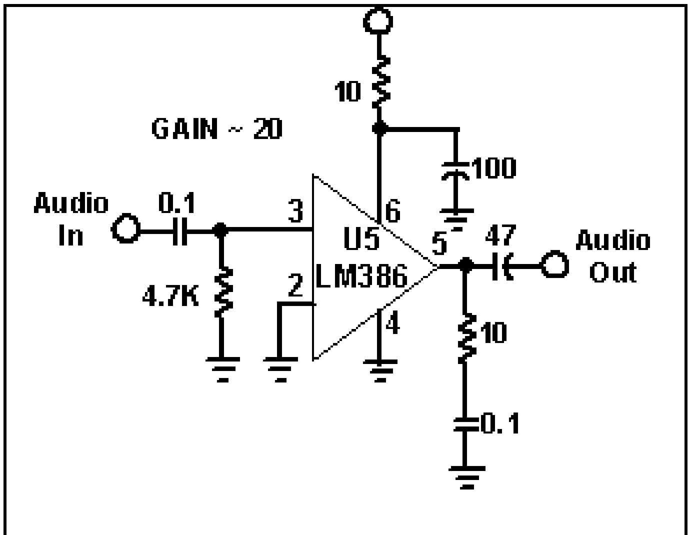


Figure 2- Audio power amplifier

For RF signal sources no further changes are needed. However if an audio test signal is to be used, the filter capacitor in the detector being tested needs to be properly selected to provide adequate filtering. The same guidelines as described in the Diode Detector Testing section apply. If a test frequency of 1 kHz is used with a R_L of 1 megohm C_L needs to be at least 0.1 µf. A ceramic capacitor is appropriate here.

The test circuit is shown below in Figure 3.

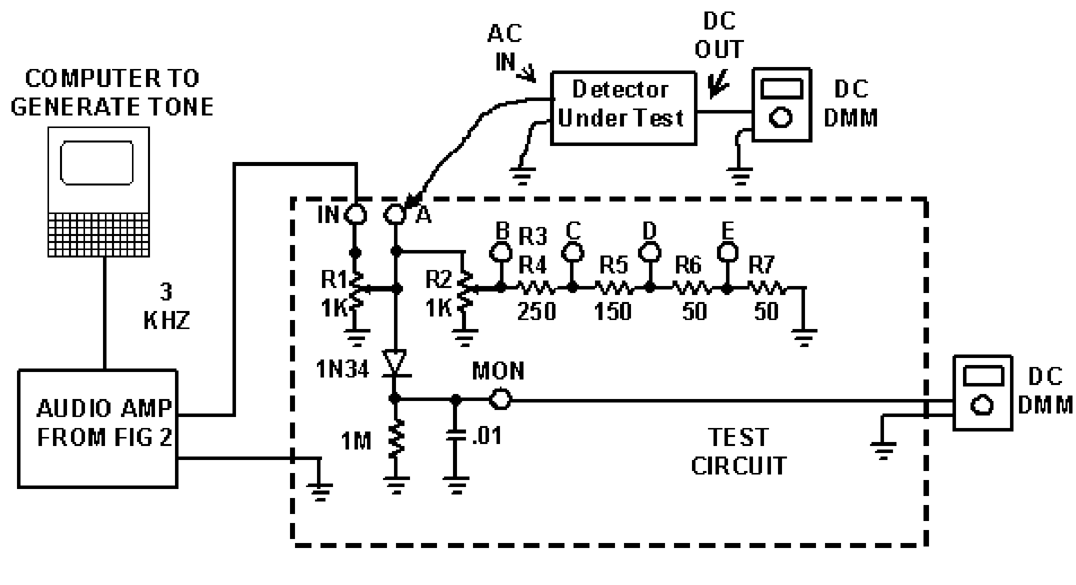
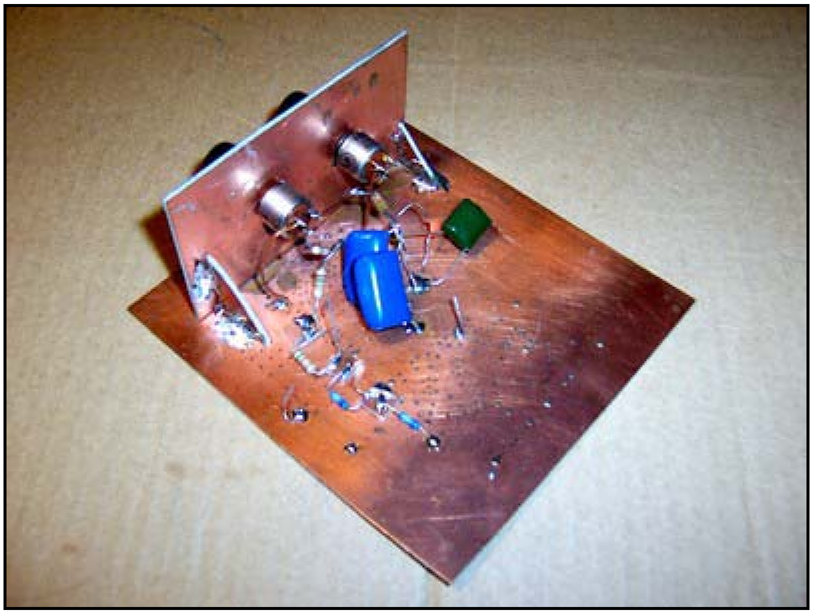


Figure 3-Compensated detector tester

Signal inputs of up to about 3 or 4V rms can be handled with ease. The input signal is reduced to a standard 1V level by adjusting potentiometer R1. The exact 1V level is assured by measuring the DC output of the detector on point A, R1's center arm. R2 then serves as a variable attenuator to reduce this 1V level to calibrated fractions of 1.0, 0.5, 0.2 and 0.1V. It has a hand-calibrated dial to allow exact level setting. Finally, fractions of R2's output are selected by a resistive divider. Taps are provided at points B, C, D and E in the ratios 1.0, 0.5, 0.2 and 0.1 times the voltage at point B. The net result of the two potentiometers and the resistive divider string is an accurately settable attenuator. That can be set to produce outputs in 1-2-5 steps from 1V down to 0.01V. Measurement accuracy is reasonably good as high as 3.5 MHz with simple construction techniques.

There are only a couple of critical components in the compensated detector tester. Potentiometers R1 and R2 must be non-inductive good quality panel-mount devices. It is best to use a 1N5711 Schottky diode for the calibration detector. The 1megohm detector load resistor R2 can be any low power 5% device. Filter capacitor C1 should be a disc or monolithic capacitor. The resistive divider chain R4 through R8 can be either 1% precision film resistors or more common carbon composition or carbon film devices. If precision resistors are used, R4 and R5 can be a single 249 ohm unit, R6 is 150 ohms, and R6 and R7 are both 49.9 ohms. If common 5% resistors are used, selection of resistors needs to be done with a good DMM on its resistance range. In this case R4 is 100 ohms and R5 is 150. Select resistors so that the series combination is within 1% of 250 ohms. R6 is likewise selected to be within 1% of 150 ohms. Finally R7 and R8 are 51ohm resistors each selected to be within 1% of the nominal 50 ohm value.

Construction is likewise fairly non-critical. Figure 4 is a photo of a prototype tester. Note that lead lengths are maintained fairly short and the components are arranged in line as much as possible to minimize coupling of the input to output. The tester could be constructed to operate up to 30 MHz, but leads would have to be kept much shorter for minimum stray inductance and shielding between sections would be needed to minimize capacitive coupling.



Prototype Tester

The calibration detector connected to point A needs to have been previously calibrated at an input of exactly 1V rms as described in the previous section. It is important to note the DC output during that calibration step and to use the same high impedance DMM whenever this test circuit is used. This ensures that the 1V level at point A is always exactly the same value.

Potentiometer R2 also needs a one-time calibration. This needs to be done after the entire tester has been built. Use as large a pointer knob as you can get and tape a piece of clean white cardboard (cut from a 3x5" index card) to the panel. Set R1 and R2 to their highest settings and make a mark at the pointer setting for R2 on the homebrew dial. Label this mark as 1.0 on the dial. Take care to mark this precisely so that you can accurately set R2 to this setting while using the tester.

Connect a fresh 9-volt alkaline battery from point A to ground. Measure the exact DC voltage at point A with a DMM. It will be typically be somewhat higher than 9 volts. Now connect the DMM to point B, R2's center arm. Adjust R2 for a reading exactly half the measured battery voltage. Mark this point on your dial and label it 0.5. Now set R2 to read exactly two tenths of the measured battery voltage. Make another dial mark and label it 0.2. Finally set R2 for one tenth the measured battery voltage, mark the dial and label it 0.1. R2's dial is now calibrated.

The compensated detector test set is now ready for use. A set of steps will be given below to indicate how it can be used to check a compensated detector or other sensitive RF detector. The signal source for this test is assumed to be an audio tone at 3 kHz generated by a pc running DigiPan software. As noted above, the pc sound does not produce a high enough output level so the audio amplifier in Figure 2 is used.

1. Ensure that the filter capacitor in the detector under test is sized appropriately for operation at 3 kHz. Guidelines have been provided above. The extra capacitance can be temporarily connected in parallel with the normal filter capacitor and removed after testing.

2. Connect the signal source, the test set and the detector to be tested as shown in Figure 5.

3. Set up DigiPan to produce a sine wave output frequency of 3 kHz and set the PC audio output level to a setting that will provide about 2V input to the test set. You can use an AC range on a DMM for this setting since it is only approximate.

4. Set R1 and R2 on the test set to their highest settings.

5. Set R1 so that the DMM on the calibration detector reads exactly the correct pre-calibrated value corres-ponding to an audio level of 1V rms at point A.

6. Connect the detector under test to point A and note its DC output reading. This is the output for 1V rms input.

7. Connect the detector under test to point B and note its DC output reading. This is the output for 0.5V rms input.

8. Connect the detector under test to point C and note its DC output reading. This is the output for 0.2V rms input.

9. Connect the detector under test to point D and note its DC output reading. This is the output for 0.1V rms input.

10. To continue testing at levels of 0.1V down to 0.01V set R2 to the 0.1 mark and repeat steps 6 through 9. Output readings will be for voltages of 0.1, 0.05, 0.02 and 0.1V input to the detector under test.

Other combinations of output voltages can be achieved by appropriate use of the R2 dial and choice of the attenuator taps A through D.

Terminating Wattmeter Test and Calibration

Several methods can be used to test and calibrate a terminating wattmeter (Ref 3). Methods described here will be limited to QRP levels though they can be extrapolated up to power levels of 10 watts or so without special equipment needs.

The first order of business is to assure that the 50 ohm dummy load in the terminating wattmeter is indeed exactly 50 ohms. If not, the power indications will be incorrect. The DMM can be used to measure the load resistance. However all is not lost if the resistance is slightly off. You can either note the inaccuracy and continue or you can recalibrate the meter if there is a calibration adjustment in it. If not you can always make up a calibration chart.

Probably the simplest method to test a terminating wattmeter is to disconnect the wattmeter's detector from the dummy load and test the detector separately. The detector can be tested at 60 Hz using the first detector testing method described above. AC voltage values for this method are provided in Table 1. Voltages for other power levels can be calculated using the formula V = SQRT(50 * P). Be sure to size the detector filter capacitor properly for 60 Hz as ahs been described above and remove the capacitor when testing is done. Reconnect the detector to the load when testing has been finished.

Table 1 – Voltage vs power

RF Voltage	RF Power
1 V	20mW
1.58V	50mW
2.24V	100mW
7.07V	1W
10V	2W
15.8V	5W
22.4V	10W

A more direct way to measure the terminating wattmeter is to use a QRP transmitter whose output power is adjustable. However do not rely on power metering within the rig since it likely is not terribly accurate. The best way is to measure the voltage across the load with a calibrated RF detector.

The simple half-wave detector shown in the Diode Detector Testing and Calibration Section above is fine for this use. Build it onto a scrap of Vectorboard using short leads and bring out 1" wires terminated in mini-gator clips for the RF input. Simply calibrate its output voltage with a known DMM and record the results in a table similar to Table 2. Additional interim levels can be added if needed.

Table 2 – Detector Calibration Table

RF Voltage	RF Power	Measured DC Out
1 V	20mW	_______V
1.58V	50mW	_______V
2.24V	100mW	_______V
7.07V	1W	_______V

10V	2W	_______V
15.8V	5W	_______V
22.4V	10W	_______V

To perform the test connect the wattmeter to be tested the QRP rig and the calibrated detector as in Figure 5.

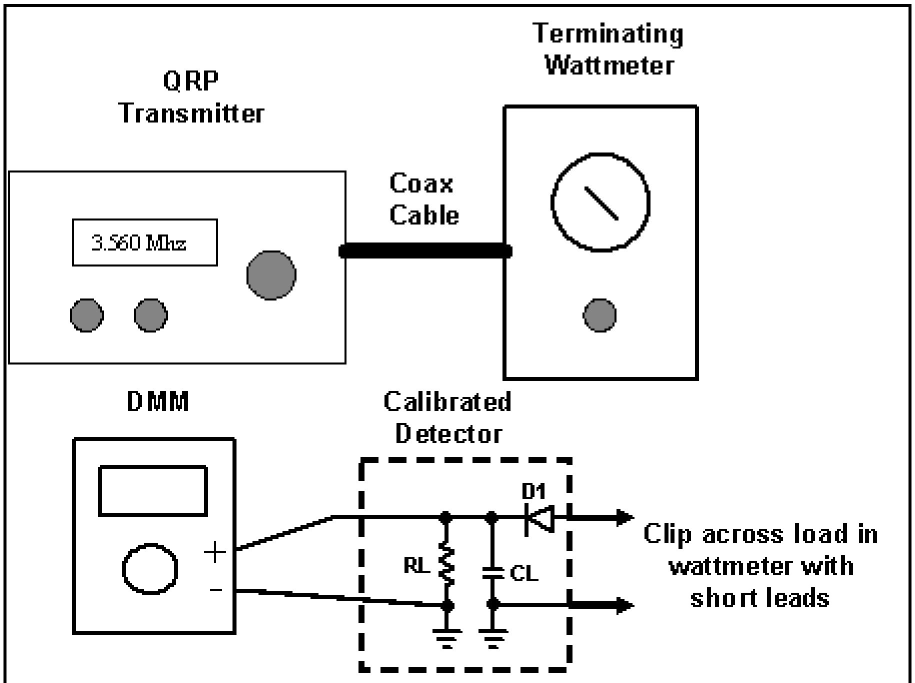


Figure 5 - Terminating Wattmeter Test Setup

Testing the wattmeter is a short and almost trivial process. The following steps detail testing from 0.5 to 5 watts. The test setup is not frequency critical so it can be done in any (or all) of the HF amateur bands.

1. Key the transmitter and adjust its output to get the DMM reading for the 0.5 watt level from Table 2. Note the wattmeter reading at the 0.5 watt input level.

2. Repeat the above step for inputs of 1, 2 and 5 watts.

Directional Wattmeter Testing and Calibration

Testing of directional watt-meters is easiest to perform using a QRP transmitter whose output level is adjustable as was used in the previous section. To measure the RF level through the meter, use either a calibrated terminating wattmeter or simulate one with a good 50 ohm dummy load and the calibrated detector described in the previous test setup.

A simplified test setup is shown in Figure 6.

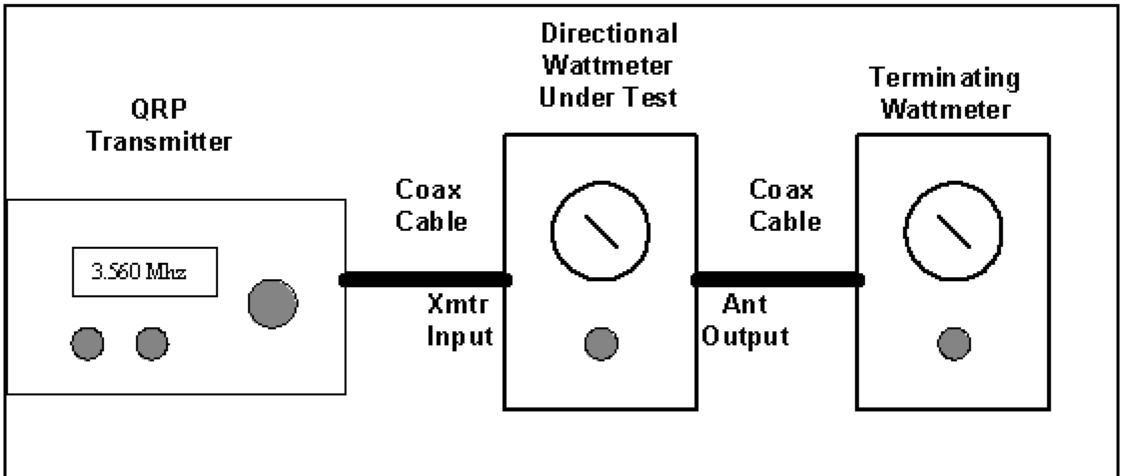


Figure 6 – Simplified Directional Wattmeter Test Setup

Testing is first done with the variable transmitter connected to the XMTR input jack of the directional wattmeter under test and the terminating wattmeter connected to the Antenna Output jack. The directional wattmeter controls are set to read forward power. Transmitter power is set successively to the desired power levels and the directional wattmeter readings are noted. Finally reverse power testing is performed by connecting the transmitter to the directional wattmeter Antenna Output jack and the terminating wattmeter to the XMTR input jack. The directional wattmeter controls are set to indicate reverse power. As before transmitter power is set successively to the desired power levels and the directional wattmeter reverse power readings are noted.

More detailed testing of a directional wattmeter will be presented in the next installment of the RF Wattmeter Cookbook where the final wattmeter design will be discussed.

References:

1. **http://mywebpages .comcast.net/hteller/ digipan/**

2. **http://www.njqrp.org/dds/ index.html**

3. Homebrewer #1, Summer 2003, RF Power Meter Cookbook, Joe Everhart, N2CX

Joe Everhart, N2CX may be contacted at 214 New Jersey Road, Brooklawn, New Jersey 08030. Joe may also be reached by e-mail at n2cx@amqrp.org

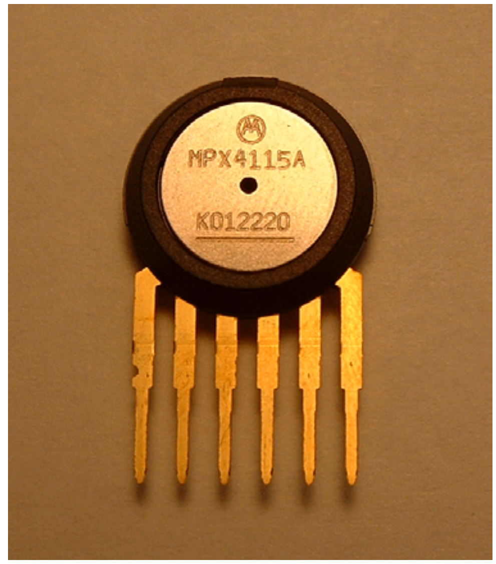
AmQRP.ORG HOMEBREWER

David Ek, NK0E

The PIC-based APRS Weather Station Project

... Part 6: Atmospheric Pressure Measurement for PIC-Wx

Adding an atmospheric pressure sensor to this ongoing fun and educational project.



PIC Source Code   **PIC Hex Code**   **PC Program**   **Calibration Spreadsheet**

Hi, everybody! Installments in my PIC weather station project have been few and far between lately, but things are going to be picking up again with this fun installment where we add an atmospheric pressure sensor to the station. By the way, thanks to those of you who've taken the time to drop me an email and tell me that you're following along and having fun. It really makes it easier to buckle down and crank new installments out when I know someone's looking forward to them. Don't hesitate to drop me a note with a question or a comment on the project. Now, let's get started by briefly reviewing the concept of atmospheric pressure.

Barometer Basics

Barometers are devices that measure atmospheric pressure. Atmospheric pressure exists because, quite simply, air has weight. The weight of the air pressing down on you from above is about 14.7 lbs/in^2 if you're at sea level, and it decreases as you increase your elevation above sea level. Atmospheric pressure is commonly expressed as some number of inches of mercury (in. Hg) that is displaced by the weight of the air. The average atmospheric pressure at sea level is usually given as 29.92 in. Hg. This is really an indirect way of specifying atmospheric pressure, though, since you have to calculate the weight of that column of mercury in order to determine the actual pressure needed to displace it. Other common units of pressure are kilopascals (kPa) and millibars (mb). Millibars are also known as hectopascals (hPa). To convert from one unit to another, you only need to know that one in. Hg equals 3.3857 kPa and that 10 mb equals 1 kPa. You can do the math.

One other thing worth noting is that, regardless of your elevation above sea level, the atmospheric pressure reported for your location is the pressure that we estimate we'd measure at sea level for that location. This is done so that all higher elevations don't show up as low pressure areas on the weather maps. The estimate can be done a number of ways, but it is usually based on the U.S. Standard Atmospherei[i], an average profile of the atmosphere developed by the government that gives average absolute atmospheric pressure for a number of altitudes (among other things). In our case, we'll calibrate our barometers against data that's already been corrected to sea level, so this problem will take care of itself.

The Electronic Barometer Circuit

David Bray has published a fairly simple design on the webii[ii] for a barometer based on the MPX4115A, and I've adapted it slightly for our use in this project. Figure 1 shows a block diagram of how we'll measure the atmospheric pressure. At the heart of the circuit is our pressure sensor, the Motorola MPX4115A. It outputs a voltage that is linearly proportional to the absolute atmospheric pressure. The MPX4115A is capable of measuring atmospheric pressure from 15 to 115 kPa and outputs a voltage from 0.2 to 4.8 V. However, the range of pressures that we'll encounter in measuring atmospheric pressure is only about 10 kPa wide, and the corresponding voltages output by the MPX4115A will occupy a range of less than a half volt. In order to widen that range, we'll employ an amplifier to increase the resolution we can obtain when we convert the analog voltage to a digital value in the A/D converter. The digital value will then be transmitted to the PIC, which transmits it to the PC for conversion to the actual pressure value (kPa or in. Hg). The PC then formats it and puts it out to the APRS network as part of a weather packet. That's it in a nutshell. Now let's get down to the details.

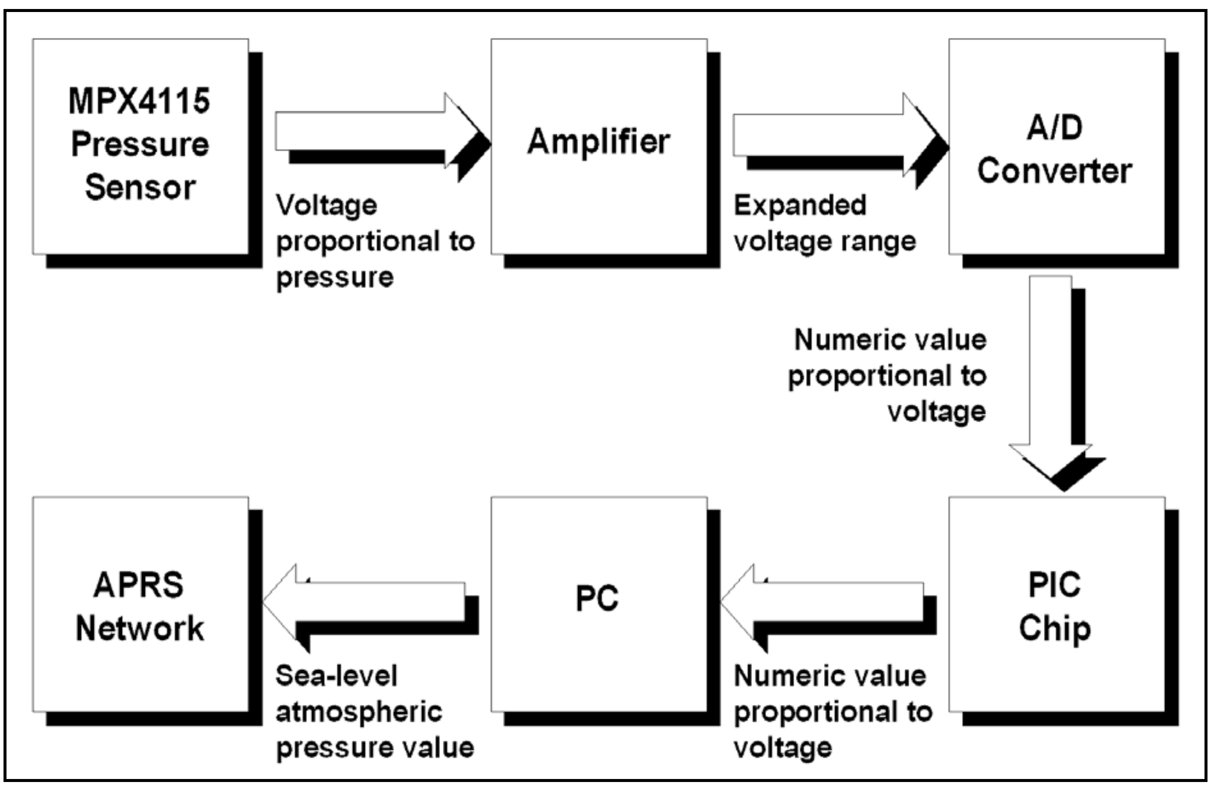


Figure 1. Barometer block diagram

The MPX4115A Pressure Sensor

The MPX4115A pressure sensor from Motorola is shown in Figure 2. Only three of its six leads are used (and other packages exist for this sensor with eight leads, in surface mount form). 5 V is provided to pin 3, pin 2 is grounded, and pin 1 (the one with the little notch in the side) outputs a voltage that is linearly proportional to the absolute atmospheric pressure. The pressure and voltage are related by

$$\frac{V_{out}}{V_S} = 0.009 \cdot P - 0.095$$

Equation 1

where V_{out} is the output of the sensor in volts, V_S is the 5 V supply voltage, and P is the absolute pressure in kPa. You can see that this is the equation of a straight line, and it is a simple matter to convert from voltage to pressure and vice versa.

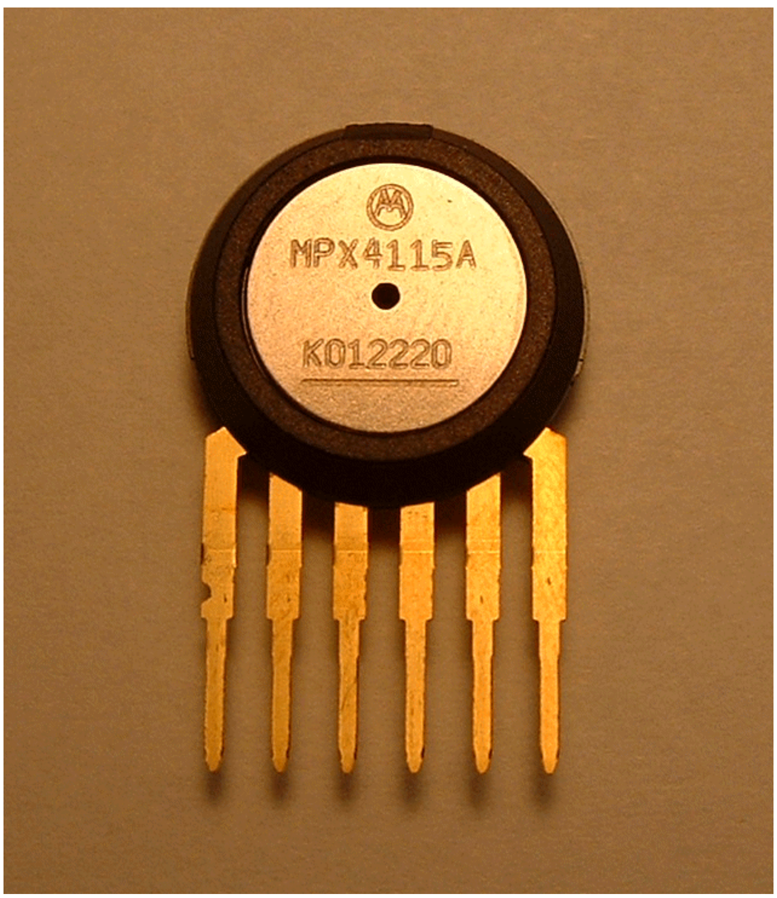


Figure 2. The Motorola MPX4115A absolute pressure sensor.

When I first received my MPX4115A, I just hooked it straight to a 5 V supply and measured its output with a voltmeter, then used the formula above to calculate the pressure. Pretty simple! But the pressure I measured was nowhere near the pressure that the National Weather Service was reporting at the airport nearby. The reason, of course, is that I'm in Colorado Springs , Colorado at an elevation of 6550 feet. My barometer is reading absolute pressure, but the NWS is reporting pressure corrected to sea level. How do we correct our barometer for sea level? David Bray used the following equation to relate absolute pressure at some elevation to that at sea level:

$$P = P_{SL}\, e^{\ln(1-6.87324^{-6}\, E)\cdot 5.256}$$

Equation 2

where *P* is the absolute pressure at elevation *E*, P_{SL} is the sea level pressure, and *E* is the elevation in feet. *P* and P_{SL} can be given in whatever pressure units you'd like, as long as you use the same units for both. If we use equation 2 for my location at 6550 ft above sea level, a sea-level pressure of 31 in. Hg corresponds to an absolute pressure equal to 24.33 in. Hg at my location. Likewise, a sea-level pressure of 28 in. Hg corresponds to an absolute pressure of 21.98 in. Hg at my location. Since this represents a reasonable maximum range for our barometer, we can use equation 1 to determine the range of voltages that will be output by the MPX4115A for this pressure range (but don't forget to convert the pressures to kPa first). Again, for my location the range of voltages output by the MPX4115A will be from 2.87 V to 3.23 V. That's not a very big range, so we'll use an amplifier circuit to amplify the output of the MPX4115A to enlarge its range and give us better resolution.

Amplifier Circuit

The amplifier circuit uses both halves of the LM358N dual op amp. Each amplifier uses negative feedback to control the gain. Pots R14 and R16 are multi-turn trimmers. They will be adjusted so that when the voltage range of interest output by the MPX4115A is input to the amplifier, the amplifier output range will be from 1.25 V to 3.25 V (the linear region for the amplifier when powered by 5 V). This way we'll get higher resolution from the A/D converter. I'm not going to explain the amplifier circuit in detail here since it's fairly simple, but drop me a note if it puzzles you and I'll help you understand it.

When I built up this circuit, I added pot R19 so that I could calibrate the amplifier. I needed a way to control the input voltage to the amplifier that's normally provided by the MPX4115A, and I couldn't think of a way to readily change the air pressure, so temporarily replacing the sensor with a pot gave me an easy way to calibrate the amplifier. My goal was to have the output of the amplifier range from 1.25 V to 3.25 V over the expected input voltage range (2.87 V to 3.23 V for me at my location). So, I disconnected the MPX4115A and used R19 to set the input voltage for the amplifier. I connected one DMM to the wiper of R19 to monitor the input voltage, and I connected another DMM to the output (pin 1) of the op amp to monitor the output voltage. Then I adjusted R19 to the upper voltage in my range of interest (3.23 V), and I adjusted R16 until the output voltage was 3.25 V. Then I adjusted R19 to the lower voltage in my range (2.87 V) and adjusted R14 until the output of the amplifier was 1.25

V. I repeated this process of alternately adjusting R14 and R16 at each end of the range several times, until the trimmers no longer needed adjustment.

Once the adjustment of the amplifier was complete, I needed a way to compute the sea-level pressure from the output voltage. First, I measured the amplifier output voltage at several input voltages over the range of interest. Then, using equation 1, I calculated the absolute pressure corresponding to that input voltage (and thus the output voltage). Then I used equation 2 to compute the corresponding sea-level pressure. The results of this exercise are shown in Table 1.

Table 1. Amplifier Calibration Data.

Amplifier Input (V)	Amplifier Output (V)	Absolute Pressure (kPa)	Sea Level Pressure (kPa)	Sea Level Pressure (in.Hg)
2.85	1.14	73.89	94.13	27.80
2.9	1.44	75.00	95.54	28.22
2.95	1.71	76.11	96.96	28.64
3	2	77.22	98.37	29.06
3.05	2.25	78.33	99.79	29.47
3.1	2.53	79.44	101.20	29.89
3.15	2.81	80.56	102.62	30.31
3.2	3.08	81.67	104.03	30.73
3.25	3.37	82.78	105.45	31.15

Once I had the data, I used Microsoft Excel to plot the sea-level pressure versus the amplifier output and compute a linear fit to the data. The resulting plot is shown in Figure 3. As you can see, the data is practically perfectly linear, and all I need to do to get the pressure from the output voltage is to use the equation for the linear fit shown on the plot. But before I can do that in my PIC chip, I need to convert the voltage to a number. Enter the analog-to-digital converter.

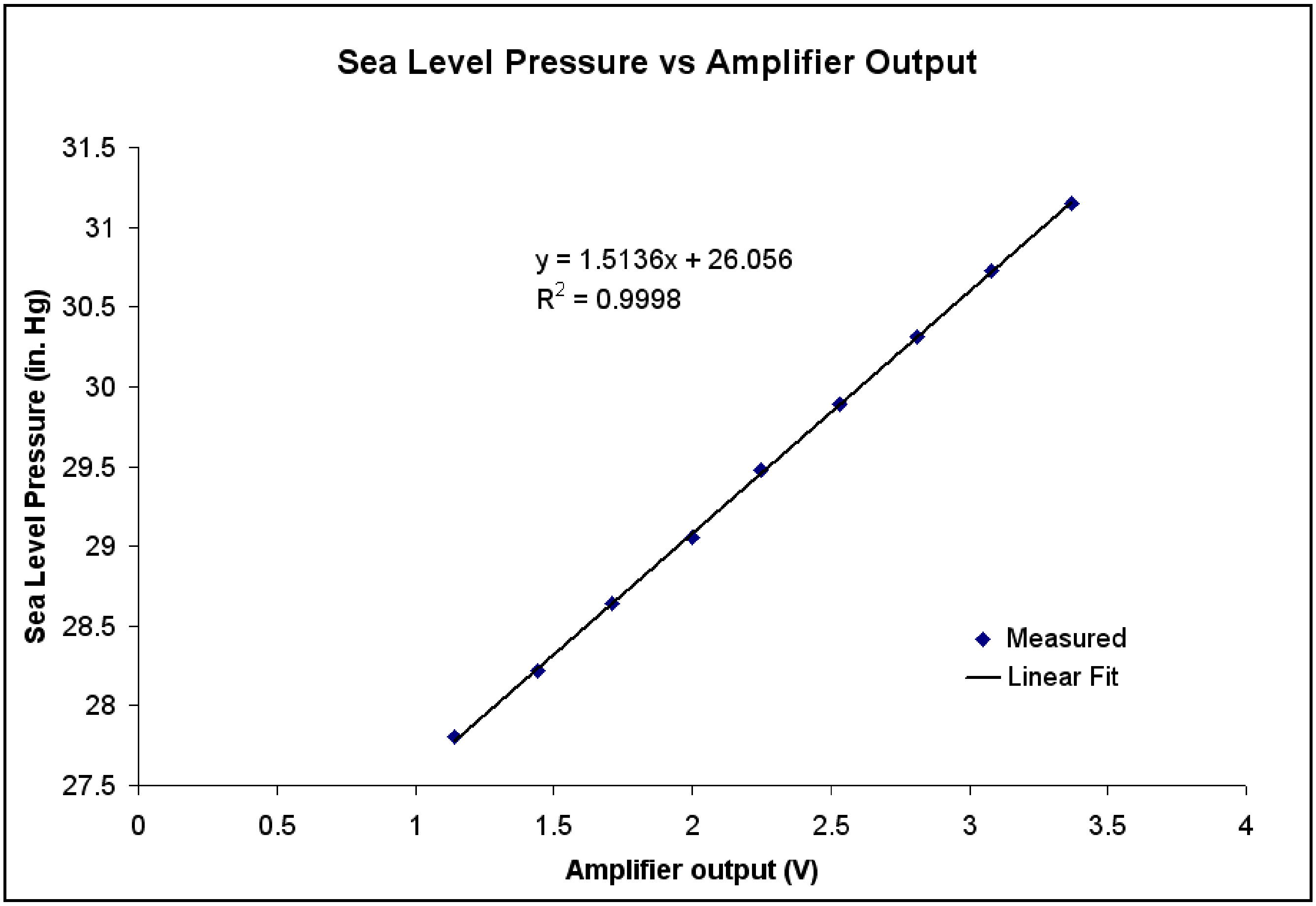


Figure 3. Calibration of Sea-Level Pressure to Amplifier Output Voltage.

A/D Converter

Unlike David Bray's design, I selected a 12-bit serial A/D converter for my barometer, the Microchip MCP3201. The MCP3201 takes three inputs. V_{REF} (pin 1) is a reference voltage that sets the upper limit for voltage measurements. V_{IN+} (pin 2) is the positive analog input and is where the actual voltage to be measured. V_{IN-} (pin 3) is the negative analog input and must be within 100 mV of ground. It can be used to cancel small-signal common mode noise that is present on both the V_{IN+} and V_{IN-} inputs. Here we simply ground V_{IN-}. Given those inputs, the output of the MCP3201 will be a 12-bit integer given by

$$N = 4096 \cdot \frac{V_{IN+} - V_{IN-}}{V_{REF}}$$

Equation 3

For this design I've set V_{REF} to be 5 V. Note that the range of the output is from zero to 4095.

The other pins of interest on the MCP3201 are CS/SHDN (pin 5), D_{OUT} (pin 6), and CLK (pin 7). These pins are used to get the MCP3201 to send the digital output one bit at a time to the PIC. CS/SHDN is used to signal the MCP3201 to make a measurement and send it to the PIC. The CLK (clock) line is repeatedly toggled between high and low to synchronize the data output between the MCP3201 and the PIC. The D_{OUT} line is used to actually send each bit. Here's how it works. CS/SHDN is held high until it's time to take a voltage reading, at which point it must be taken to ground. The MCP3201 will then begin to sample the voltage inputs as soon as the CLK line is taken from low to high, and the sample will end in the falling edge of the second clock. The D_{OUT} line is set to high or low on the falling edge of each subsequent clock cycle to signal the value of the bit being sent. After all twelve bits are sent (the most significant bit sent first), the CS/SHDN line must be taken high again before another measurement can be made.

It should be fairly obvious at this point that once we have the number from the MCP3201 it's a piece of cake to compute the pressure. Simply use equation 3 to determine the voltage from the number received from the MCP3201, and then use the formula given on the graph in Figure 3 to find the sea-level pressure.

Along with the usual PIC code and PC software, you'll find a spreadsheet available for downloading as part of this installment that automates the process of creating your calibration formula and generating the calibration numbers that you'll need to stick into the PC code. Numbers colored in red in that spreadsheet are the values you'll need to enter. You'll enter the elevation above sea level for your station and the minimum and maximum sea level pressure values over which you'll want to measure (28 and 31 in. Hg are suggested). The spreadsheet will then tell you the minimum and maximum output voltages of the MPX4115 pressure sensor for which you should calibrate your amplifier. Once calibrated, you can enter the input and output amplifier voltages like what are in Table 1, and the spreadsheet will calculate the M and B values (slope and y-intercept) that you'll enter into the PC software.

PIC Code

As the schematic indicates, the MCP3201's CS/SHDN line is connected to RB5 on the PIC, the DOUT line is connected to RB6, and the CLK line is connected to RB7. The PIC code that was needed for the barometer was fairly simple. First, I added a handler for the 'p' command so that we had some way of signaling the PIC to get the pressure and tell us what it was. Second, I added subroutine TellPressure to do the actual communication with the MCP3201. TellPressure starts the measurement by taking RB5 low to signal the MCP3201 to start measuring. Then RB7 (the clock) is cycled twice to complete the measurement. After that, subroutine GetPressureByte is called twice to get the data. Before calling GetPressureByte, the FSR register is loaded with the address of the register in which the data should be stored, and the W register is loaded with the number of bits to get. The first call to GetPressureByte gets the most significant four bits (into the hi register), and the second call gets the remaining eight bits (into the lo register). Once the lo and hi registers are loaded, SendAsciiNum is called to send the pressure value to the PC via the serial port.

If you're not familiar with their use, the FSR and INDF registers are special registers that give you the ability to do indirect addressing. If you address the INDF register with an instruction, you are really addressing the register whose address is stored in the FSR register. In the GetPressureByte subroutine, you'll see that the value of RB6 (the data line from the MCP3201) is always written to bit 0 of INDF, and then INDF is rotated to the left to make room for the next bit. By preloading the FSR register with the address of either the lo or hi register, the GetPressureByte subroutine can write to either register, making it more versatile. Another use of the FSR and INDF registers is if you need to perform some action on a range of contiguous registers. Simply by incrementing the FSR register you can use INDF to address each register in the range sequentially.

Data Acquisition

Once I had the circuit sending pressure data over the serial port, I did a quick modification to my PicWx PC software so that it would get the data and use the formulas given above to compute sea-level pressure. I also had it write the raw A/D output and the computed sea-level pressure out to a data file along with the date and time of each measurement. I set the software to read the pressure every five minutes and let it run for a couple days.

While I was accumulating pressure data from the circuit, I went to the web in search of National Weather Service pressure measurements that I could compare to the pressures I was measuring. I discovered that a two-day history is available for practically every reporting location. To find this data for your location, go to http://weather.gov/ and enter your city and state in the space on the left side of the page to get to your local forecast. On the web page for your local forecast, you'll see an area labeled "Current Conditions," where the current weather conditions and the place they're being measured are reported. Just below that is a link to the "2 Day History". Click that link to see hourly observations for the previous 48 hours.

After I had collected a day's worth of data, I was eager to see how well it compared to the official NWS data, so I used Excel once again to make a graph, shown in Figure 4. As you can see, the agreement between my measured data and the NWS data is excellent. I had thought that I'd need to adjust the calibration formula once I compared to real measurements, but I can't see how I could improve the agreement appreciably.

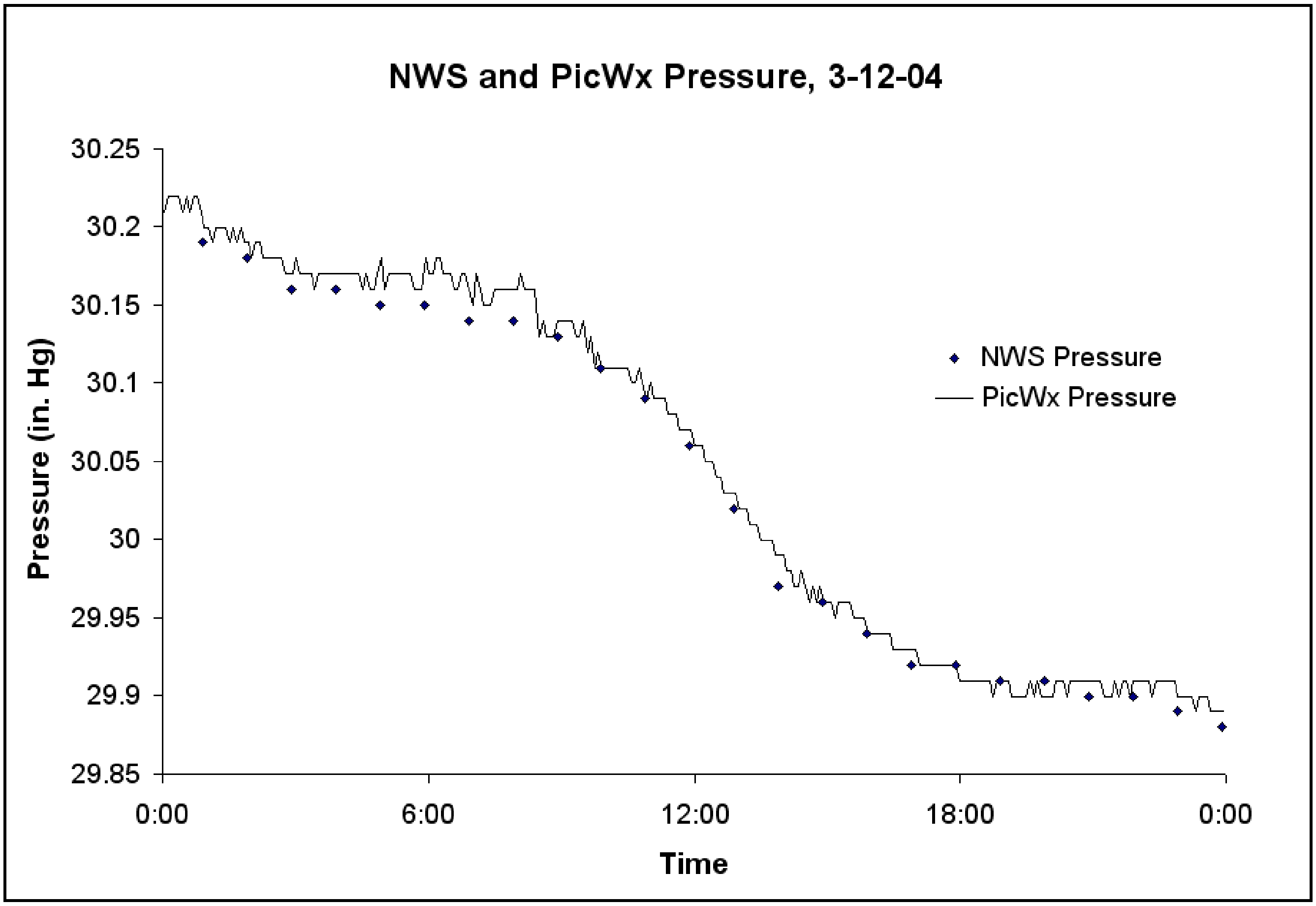


Figure 4. Comparison of PicWx measured pressure with NWS measurements

PC Software

I updated my PicWx PC software to include the measurement of barometric pressure. You can see the latest version in Figure 5. Figure 6 shows the settings dialog box, and this has changed quite a bit since the last version. First, now you can select among various units for displaying your data. I received several requests for metric units, so now you Canadians and Europeans can hopefully see the readings in familiar units (if I missed some, let me know). The settings screen also includes the calibration values for the barometer. Simply take the values created for you in the spreadsheet I told you about earlier in the article, and transfer them to the fields shown in this dialog box. Nothing to it, right? Finally, I enabled the logging feature so that you can capture the data you measure to a file for later examination. Simply check the box, and the data will be logged into a file named picwx_XXXXX.log (where XXXXX is “temp”, “hum”, “pres”, “dewpt”, or “wind”) in the same directory where you installed the PicWx software (probably \Program Files\PicWx). It’s very easy to import the contents of these files into a spreadsheet for graphing.

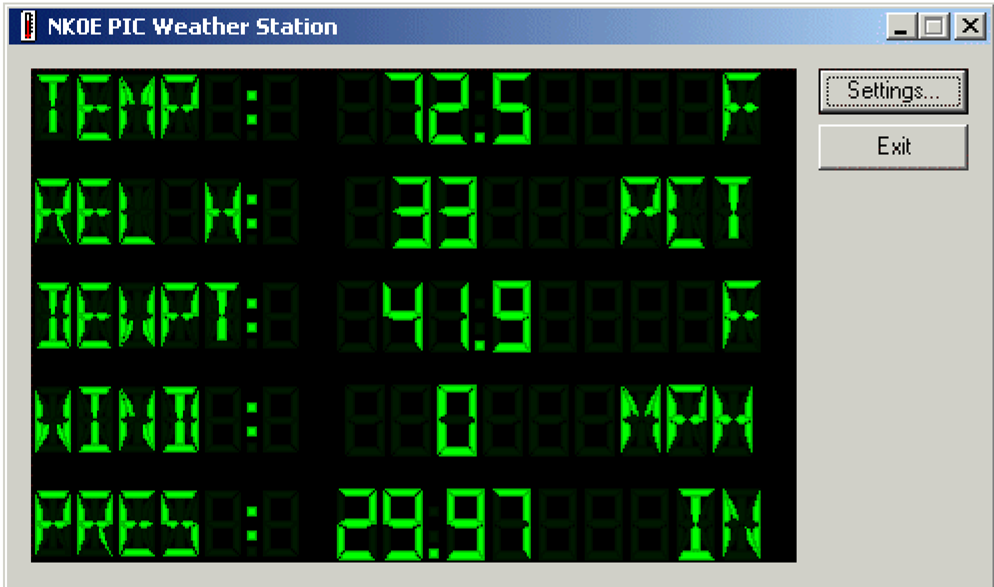


Figure 5. The PicWx PC software main window

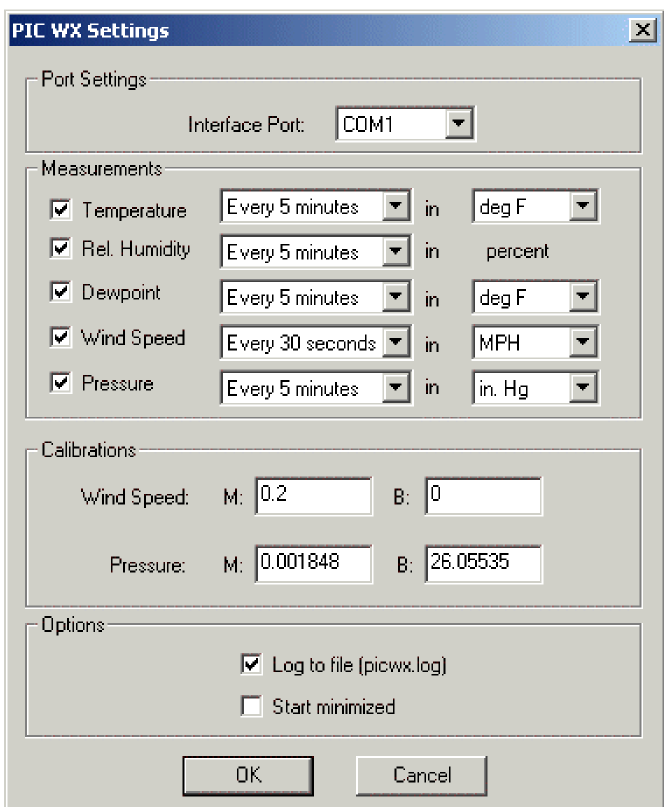


Figure 6. The PicWx PC Software settings dialog box.

Wrap-up

Be sure to grab the Microsoft Excel spreadsheet along with the software for the PC and the PIC from the NJQRP web site, so you can quickly and easily create your own calibration data. By the way, if you're not lucky enough to have access to Microsoft Office, there is a free alternative called Open Office that's very good and can handle files created by Microsoft Office. It's a big download (over 50 MB), but it's a good alternative to shelling out big bucks for Microsoft's package. You can find it at www.openoffice.org.

That's all for this installment. I'm keeping the topic of the next installment a secret (even from myself, until I figure out what it's going to be about). Keep those cards and letters coming!

73, Dave NK0E

i[i] U.S. Standard Atmosphere, 1976, U.S. Government Printing Office, Washington , D.C. , 1976.

ii[ii] http://davidbray.org/onewire/barometer.html

David Ek, NK0E may be reached by mail at 5605 Oro Grande Drive, Colorado Springs, Colorado 80918, or by email at david.ek@earthlink.net

AmQRP.ORG HOMEBREWER

Richard Meiss, WB9LPU

A Homebrew PAC-12 Antenna

WB9LPU takes the basic design of the very popular KA5DVS "PAC-12" portable multi-band vertical antenna and makes it sturdier and more easily assembled in the field.

This is an adaptation of the PAC-12 vertical antenna designed by James Bennett, KA5DVS, as published in the QRP Homebrewer #8 (April 2002). I followed the basic plans for the overall construction but made a few changes in the details. These were:

1. Use of 5/16" aluminum rod instead of quarter-inch for the lower two sections.

2. Use of couplings made of three-quarter inch aluminum rod stock, with thumbscrew locking, instead of threaded rods and couplers. (I have trouble making accurate threads on rods.)

3. Terminated the coil windings with screws and solder lugs so that it would be easier to remove turns during initial tuneup.

4. Added a provision for the attachment of guy ropes.

5. Added a bracket to the base unit for attaching a SWR bridge.

Here are some pictures of various aspects of the construction, and the results of some preliminary measurements of the antenna's characteristics.

The first photo above shows all of the antenna parts prior to setting it up. Each radial wire is wound on a separate wooden spool to avoid tangling. The rectangular object at the lower right is a clamp for attaching a SWR bridge to the base of the antenna.

The next picture shows all of the antenna parts packed in a small toolbox. It came from Sears Hardware and was described as an “underseat” toolbox for pickup trucks. It is big enough to hold the antenna rods, three loading coils, three 15-foot radial wires, a small camera tripod, a mounting clamp, and a small SWR bridge.

Photo 2

The next photo is of the three loading coils. They are for 40, 30, and 20 meters. The 40-meter coil uses a three-quarter inch riser, while the other two use half-inch tubing. The end-caps I found were not flat, so I turned a flat surface on the lathe. I terminated each end of the coil windings with a solder lug held by a screw threaded into a tapped hole in the vinyl riser in order to make removing turns easier. A short wire link connected to coil to a small piece of brass sheet stock that was drilled to accept the mounting screw that went through the end-cap and into the metal couplers. The upper coupler of the 20-meter coil is show with the whip antenna attached, and the blue plastic disk, which slips over the end of the coupler, is used to attach guy ropes to stabilize the antenna in windy conditions. The antenna rods are held in the couplers by brass thumbscrews whose ends were rounded off to prevent marring the rods.

The base of the antenna can be attached to a clamp or photo tripod, and a clamp is provided for attaching a small SWR bridge. RF connection is via a BNC connector, and the radial wires are attached by the thumbscrew.

Photo 3

The base, set up for use, is shown below.

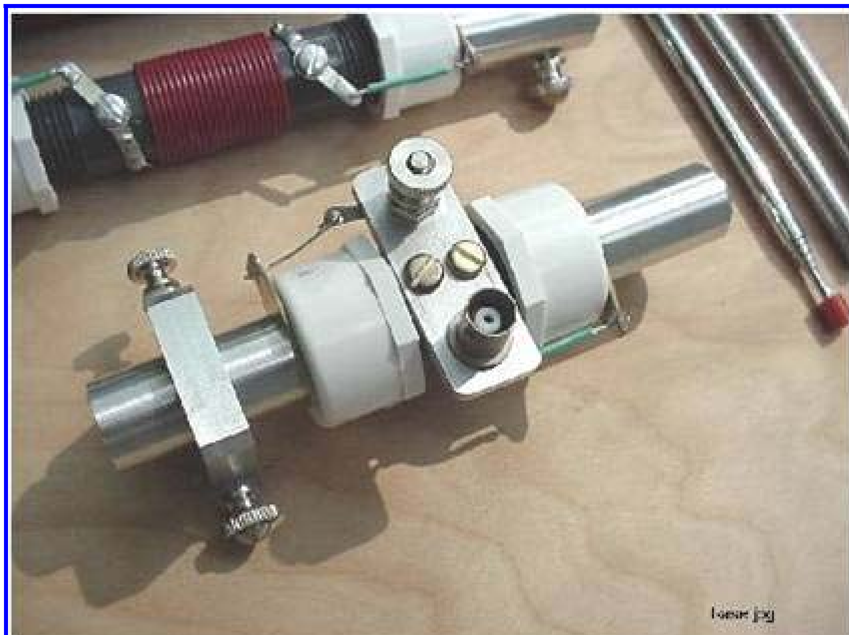


Photo 4

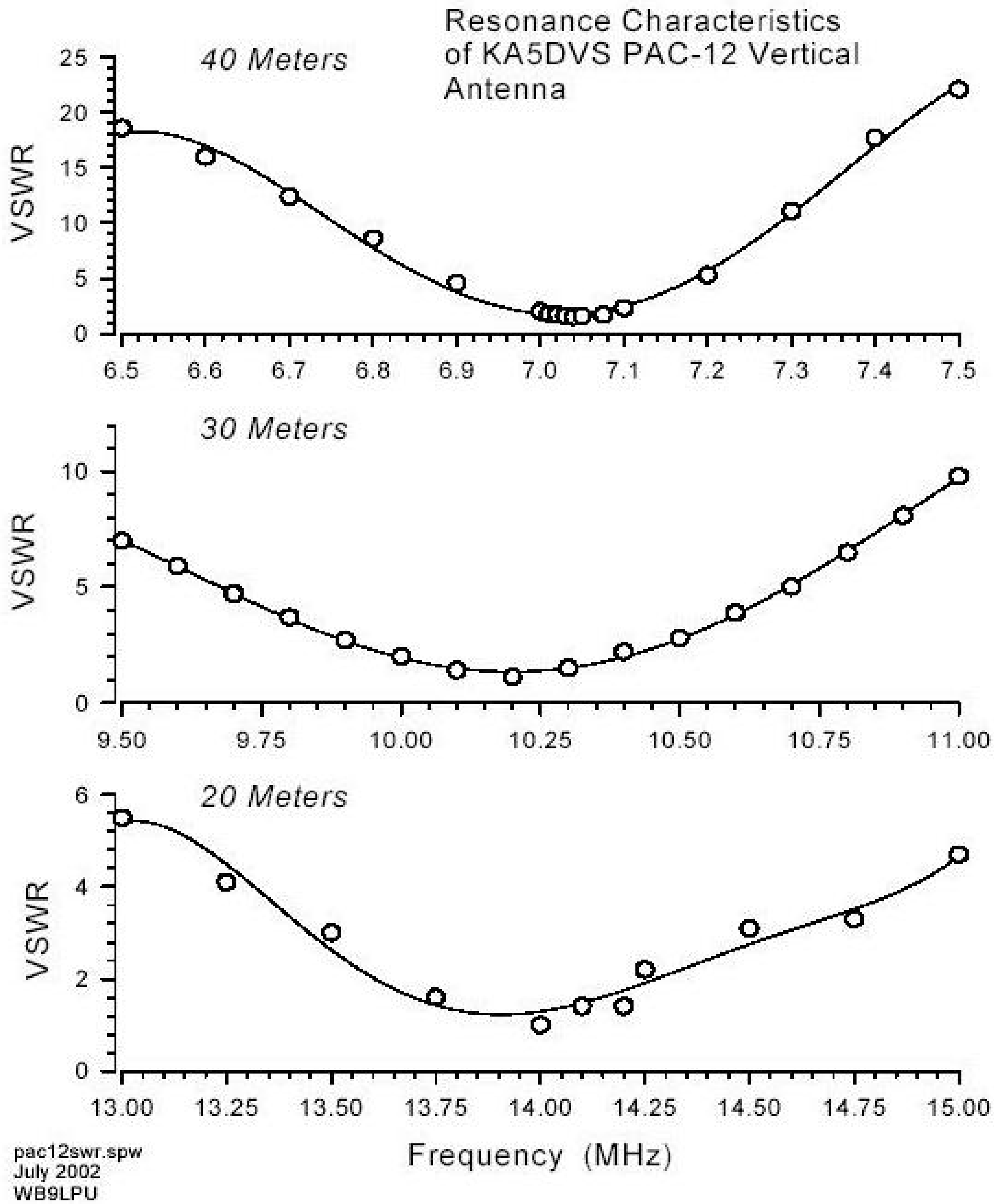
Here is the entire antenna, with the 40 meter coil in place. The whip is not extended in this view. With the coils as presently wound and trimmed, the antenna can be brought to resonance on all three bands with a slight reduction in the full extension of the whip.

Photo 5

Here is a “picnic table ham station” showing the antenna connected to a K1 transceiver. In the limited time that I have done on-the-air testing, my best distance on 40 meters is from central Indiana to South Carolina during daylight hours.

Photo 6

Below are some measurements of the antenna's performance, made with the Vectronics version of the MFJ 259B antenna analyzer. There were 3 radial wires, each 15 feet long, connected to the antenna base. The tuning is broad enough that the entire CW portions of the bands can be covered without retuning.



The antenna can be completely set up and tuned in five minutes or so. When time permits, I intend to make a slider-tuned loading coil to eliminate the coil change when going from band to band.

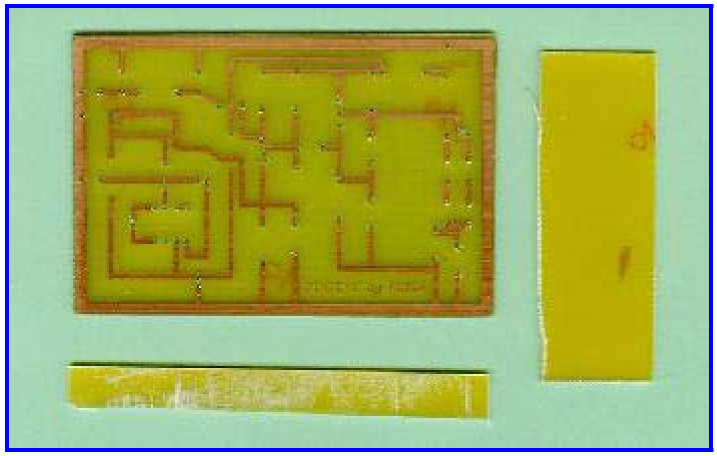
Building this antenna has been an enjoyable project, thanks in large part to the clear and detailed article by James Bennett, KA5DVS, in the QRP Homebrewer. My thanks and congratulations to James and the editors of the Homebrewer.

Richard Meiss, WB9LPU may be reached by mail at 2626 Parkwood Drive, Speedway, Indiana 46224, or by email at igeq100@iupui.edu .

Etching PC Boards at Home

Robert 'RC' Conley, KC5WA

Here's how RC followed the lead of AB5XQ, and improvised a little along the way, to homebrew a pc board for the Pixie 2 QRP Transmitter. With just a handful of parts from some local suppliers, you too can make a great-looking pcb for that next project!



Bill did it! Yes, Bill Holt AB5XQ really did a number on me. Let me explain.

I met Bill Holt, AB5XQ at Arkiecon 2003. In one of those eye ball discussions we started talking about the Side Kick (Manhatten Construction) Kit. Well Bill purchased one, took it home and from the schematic made a PCB using the Express PCB layout package, which by the way is a free download. Now I already had a Side Kick in progress before Arkiecon. So when I ran into a problem I e-mailed Bill asking if he would consider making me a board also, before I dropped my Side Kick Kit into the recycle bin. Well Bill made a PCB for me. After looking at this nice PCB, I went to Express PCB, downloaded their program and started working with it and found how simple it was to take a schematic and lay out a nice PCB.

Okay next step ... Bill, What do I need to do to make my own printed circuit boards? Huh? Bill, Huh?

After many e-mails back and forth between my self and Bill, and after bouncing my thoughts off the School M'arm (My Sister Becky) who persuaded me for once not to "make do" but purchase EXACTLY WHAT'S NEEDED to do the job, this how it went ...

From Mouser Electronics

PC BOARD ETCH SYSTEM stock #524-22-394 for $79.04 5 liter tank, no heater
-----(allied electronics pro kit, 7 liter tank with heater was $163.67)

The Kit consists of the 5 L (2.75" x 12" x 10") tank a stand that may be screwed down the air pump, 4' of tubing, a air adjustment wand and a spare heater grommet to prevent the heater from touching the tank sides. The tank is not calibrated, so this was done using a 2 cup (500ml) measuring cup. each division is 500ml. approx 1".

ETCH SOLUTION HEATER stock #524-22-392 for $16.84
-----(Allied Electronics did not sell a separate heater)

Notice the heater also comes with a stand off grommet, and is calibrate in 5 degree steps to 100 degrees

ETCH RESISTANT INK PEN stock #524-22-220 for $2.48
-----(Allied Electronics Pen $3.86)

Yup! It sure does look like a Sharpie.

From Allied Electronics

I WISH TO PUT A NOTE HERE:
The chemicals supplied in the following kits are supplied by M G CHEMICAL You may view a video on making PCB's on their web site. There is also a TECHNICAL ASSISTANCE HOT LINE 1-800-201-8822 or a FAX 1-800-708-9888 number for your use.

PHOTOFABRICATION KIT 416-k stock #661-0310 for $41.83-----(a complete kit with etchant and developer and boards)
-----(Did not find this at Mouser Electronics.)


Kit includes Developer (500ml), Ferric Chloride etchant (500ml), Developer tray, foam brushes, gloves and presensitized boards. 500ml is really not very much (2 cups) and unless your working in a flat tray very awkward to work with. So I purchased the Sodium Persulphate 1 kg (2.2 lbs) and mixes up 4 liters of etchant.

EXPOSURE KIT 416-X stock #661-0315 for $32.12
-----(Did not find this at Mouser Electronics.)

The kit includes an exposure lamp with mounting brackets and the clear acrylic panel to hold the transparency in place (shown with blue protective film still on it).

SODIUM PERSULPHATE stock #661-5019 for $24.64.-----(2.2lbs crystals 1kg makes 4 L of etchant. cost effective.)
-----(Did not find this at Mouser Electronics.)

From Walmart

SAFETY GLASSES $3.00 (YOU WEAR OVER YOUR GLASSES)

RUBBER GLOVES

Playtex Rubber Gloves $1.97

Disposable Gloves(LATEX??)

From Pampered Chef PLASTIC (Nylon) TONGS $5.75 (FOR HANDLING THE PCB's IN SOLUTION)

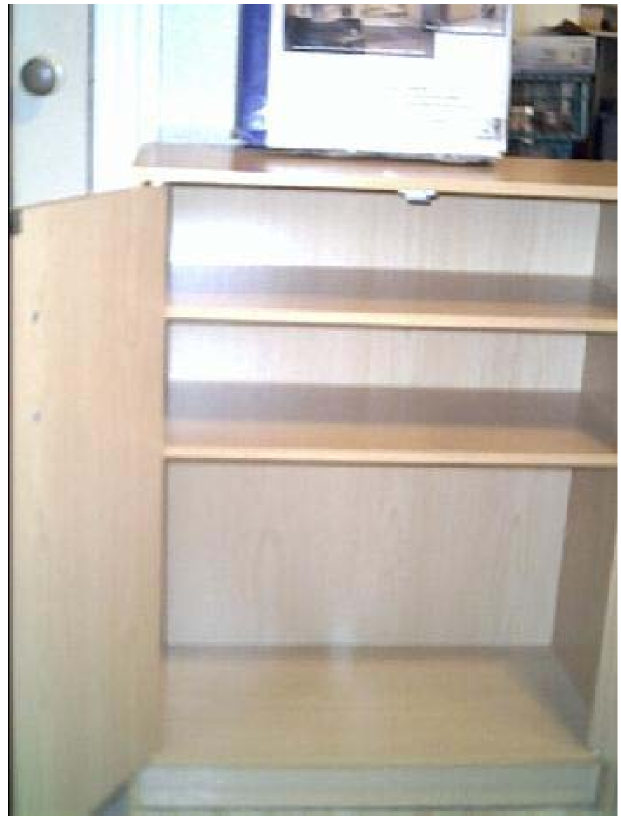
From OFFICE DEPOT

TRANSPARENCIES FOR MY LEXMARK INK JET PRINTER

A 25 sheet pack of 3M Multipurpose transparency film, Stock # 5112559820 $21.99.

From LOWES
A CABINET TO LOCK EVERYTHING AWAY IN (so I doesn't kill the nosey cat)
by Closet Maid (24"W X12"D X 31"H), ample room to store everthing away PROPERLY.
$29.88

The carton, so you know what to look for.



Assembled and ready for use, but I want to install a lock and hasp also warning signs.

Now I know it won't keep anyone out that is seriously intent on getting into it, But it should keep my very courious 5 year old granddaughter "Raven" and "Smokey" the cat from getting into harms way, Everything is complete except signs. All paper work and instructions are to be contained in a 3 ring binder for referance.

-----NOTES-----

#1 - I was given the impression from Mouser that they did not handle hazardous chemicals like etchants.

#2 - The Photofabrication Kit comes with 500ml of ferric chloride 500ml is 2 cups of liquid and is about 1" deep in a tank that measures 2.75" x 12" x 10" not enough to do the job required)

#3 - No minimum order at Mouser, $25 minimum at Allied

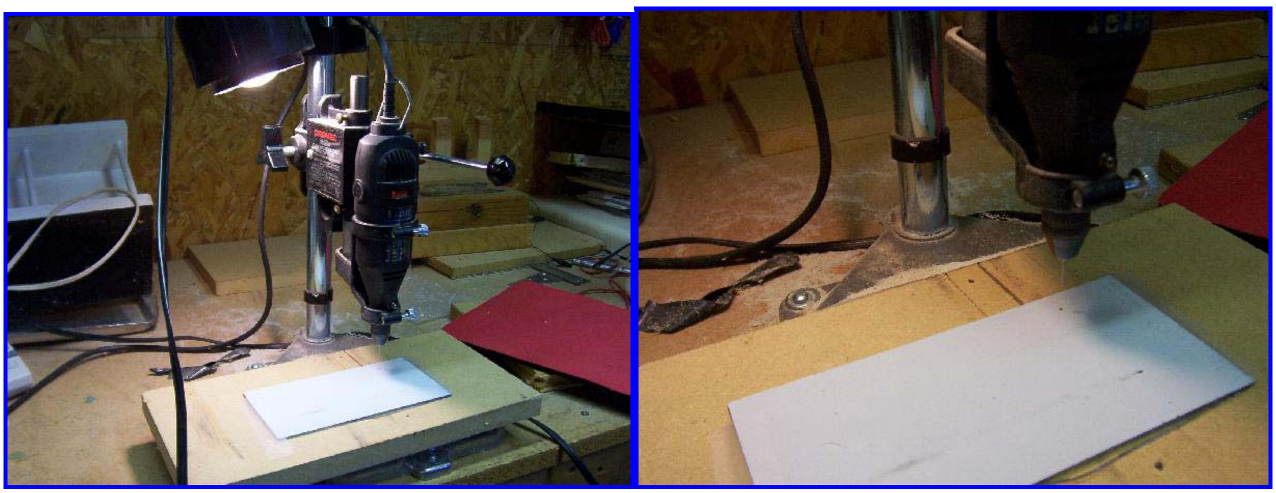
That is the set up that I have ordered and will be installing at the KC5WA home QTH.

~~~~~~

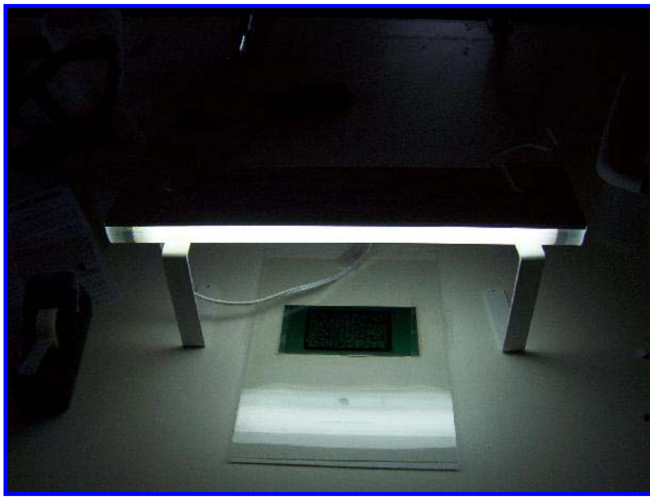
~~~~~~

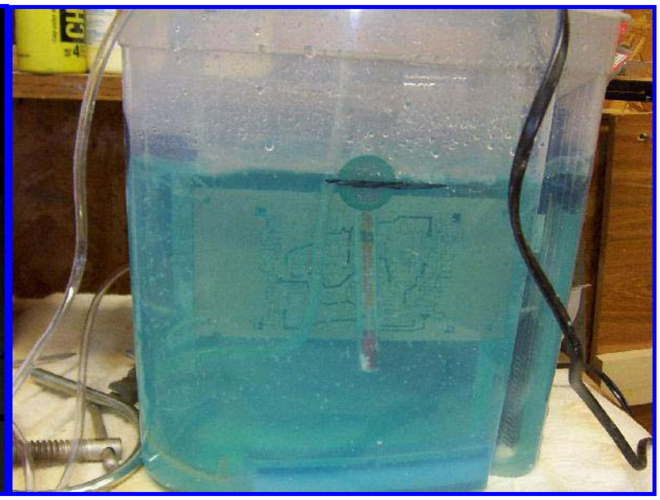
Let's take a peek at what Bill Holt AB5XQ uses at his home QTH

Here we are able to see the exposure kit, developer tank, a pre-sensitized board (in the black plastic bag) foam brush and SAFETY GLASSES "WEAR THEM"



Here is Bill's Dremel Drill Press for drilling holes after the PCB has been etched.

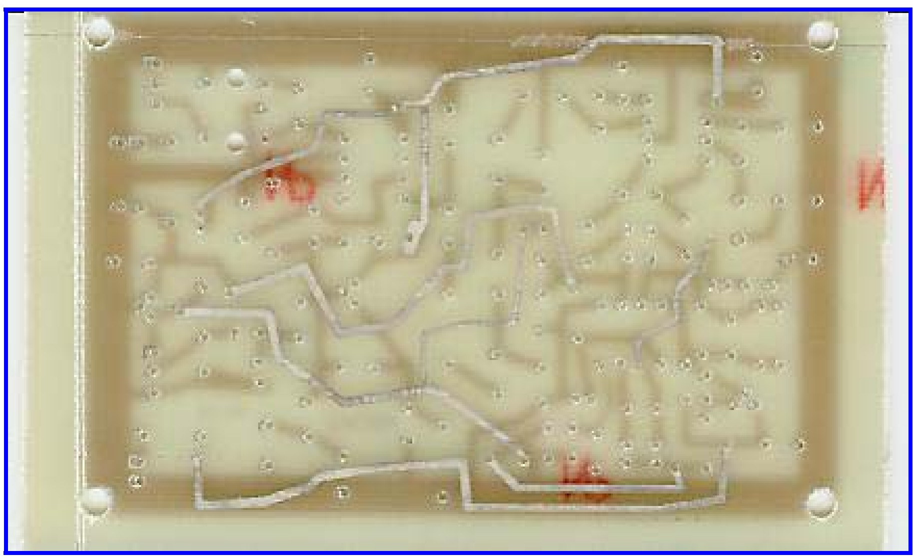


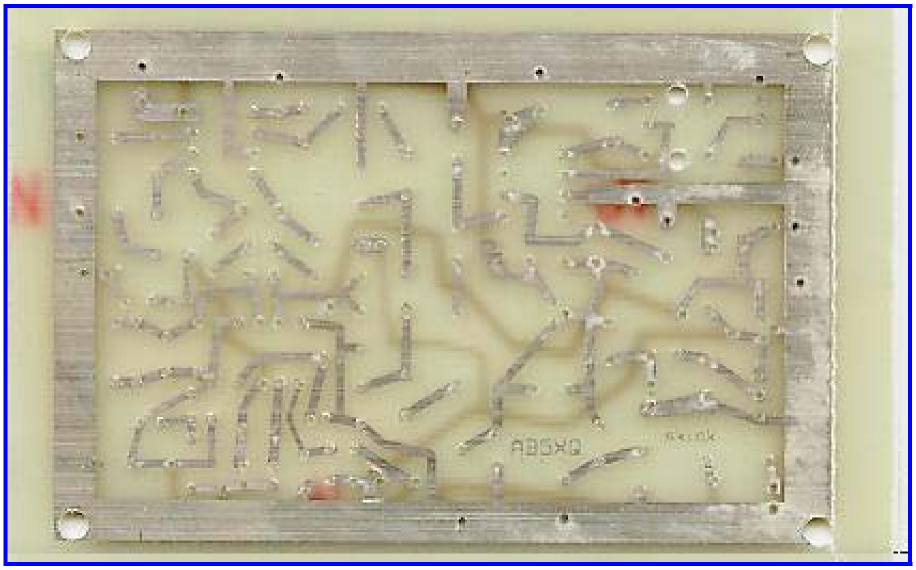


Left photo is the exposure kit and the photo on the right shows the sodium persulphate solution used for etching, notice it's opaque making it easy to see when the etching process is completed. not shown are the rubber gloves and plastic tongs needed for removing PCB's from the tank.

SAFETY NOTE: ETCHANTS ARE ACIDS DO NOT DISCARD DOWN THE SINK DRAINS UNLESS YOU WISH TO REPLACE YOUR PLUMBING. STORE IN PLASTIC CONTAINERS ONLY

This is the Side Kick board Bill made for me.



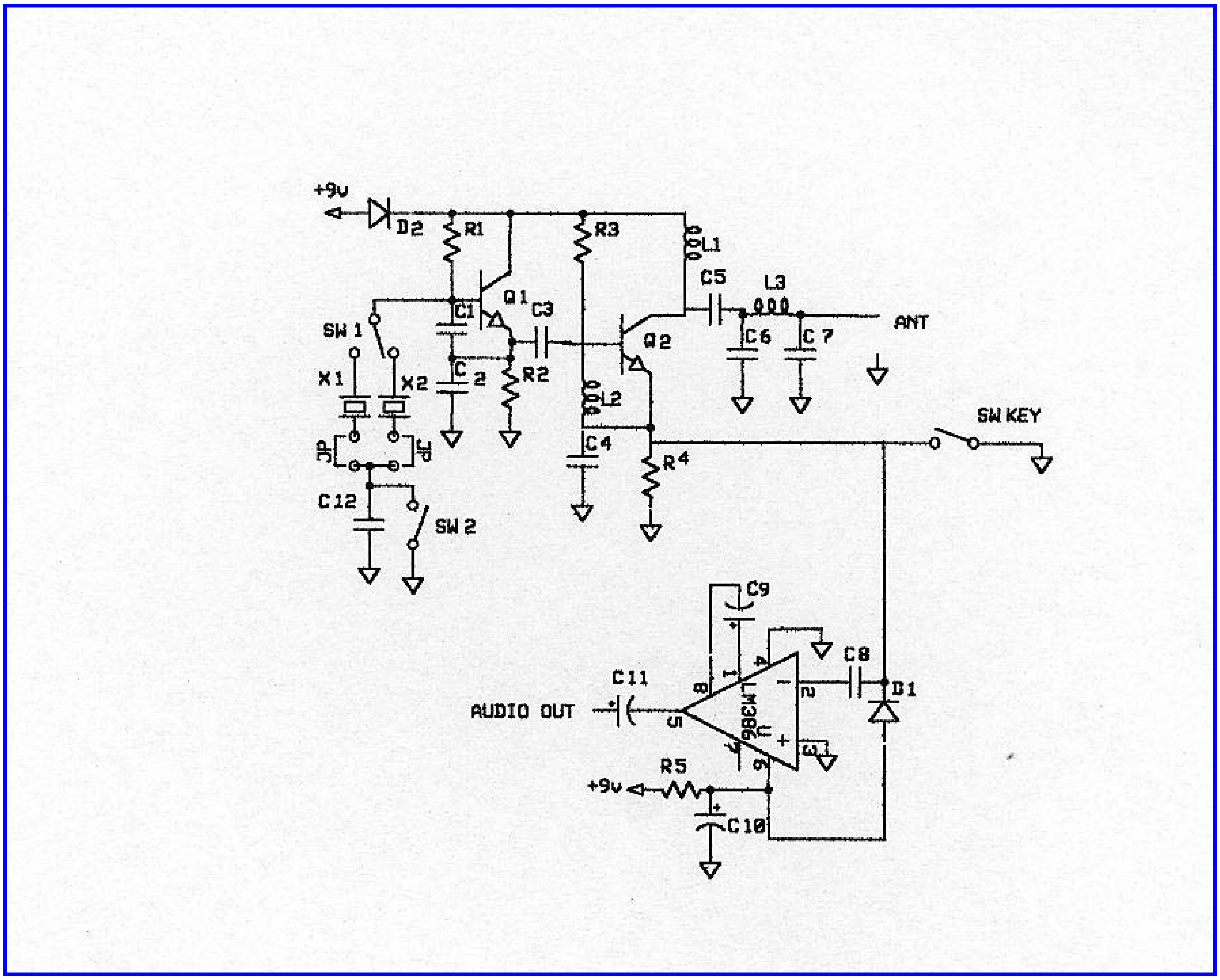


~~~~~~

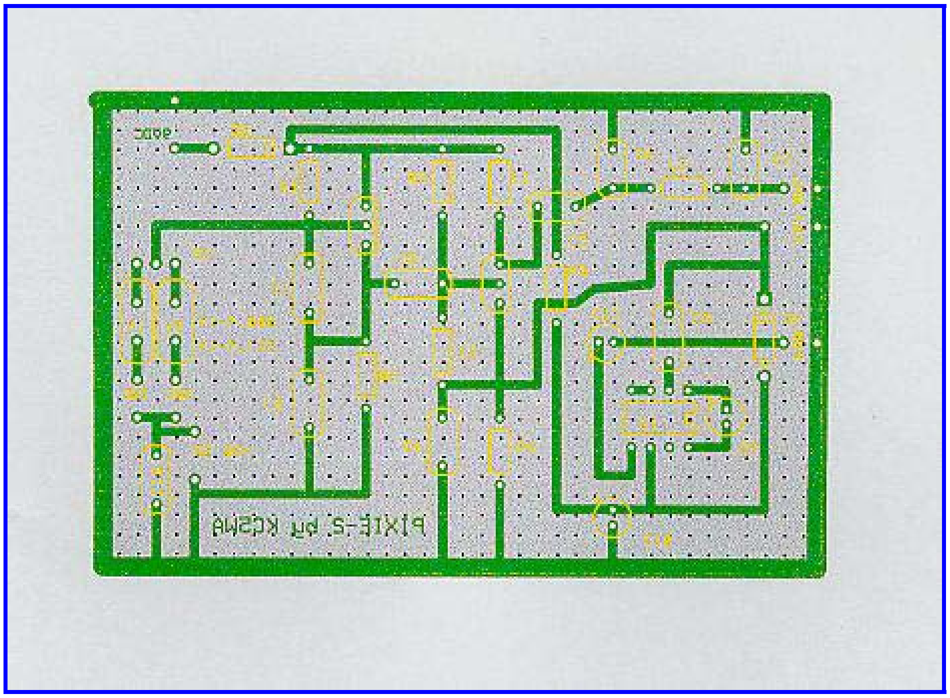
Setting up my etching facility is basically complete. Now using the FREE program that I downloaded from Express PCB earlier to layout a PCB. I am going to ATTEMPT to etch my own PCB's for a QRP transceiver and show the results here.

~~~~~~

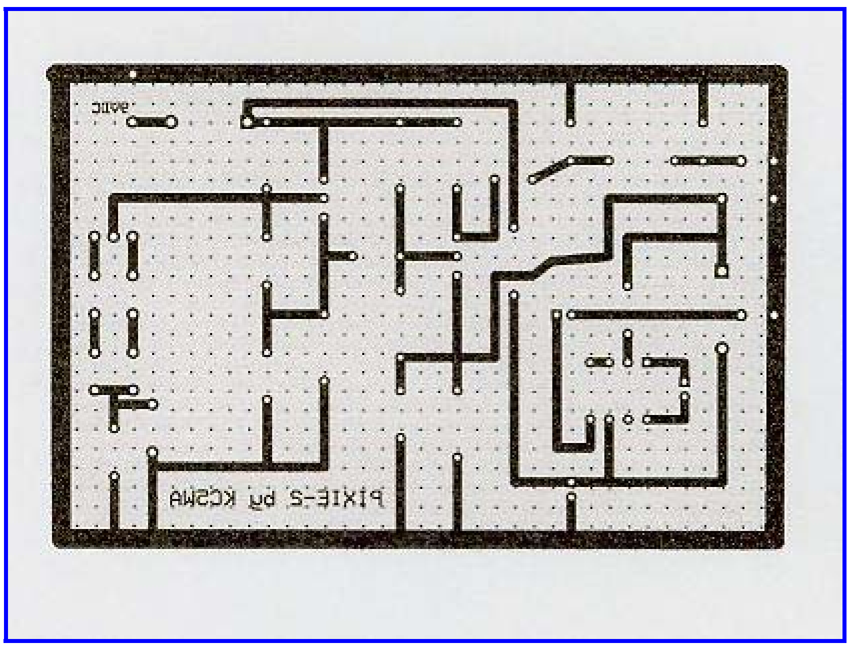
Okay! I've decided after careful consideration to etch a Pixie 2 transceiver using the FREE program from Express PCB . Let's see what I have ready.



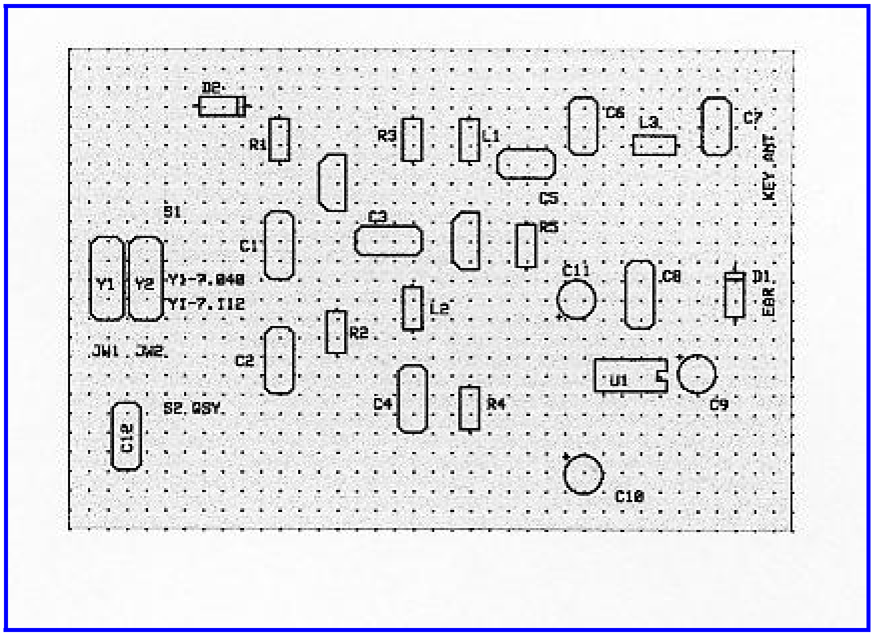
Here is the Pixie 2 schematic I used.



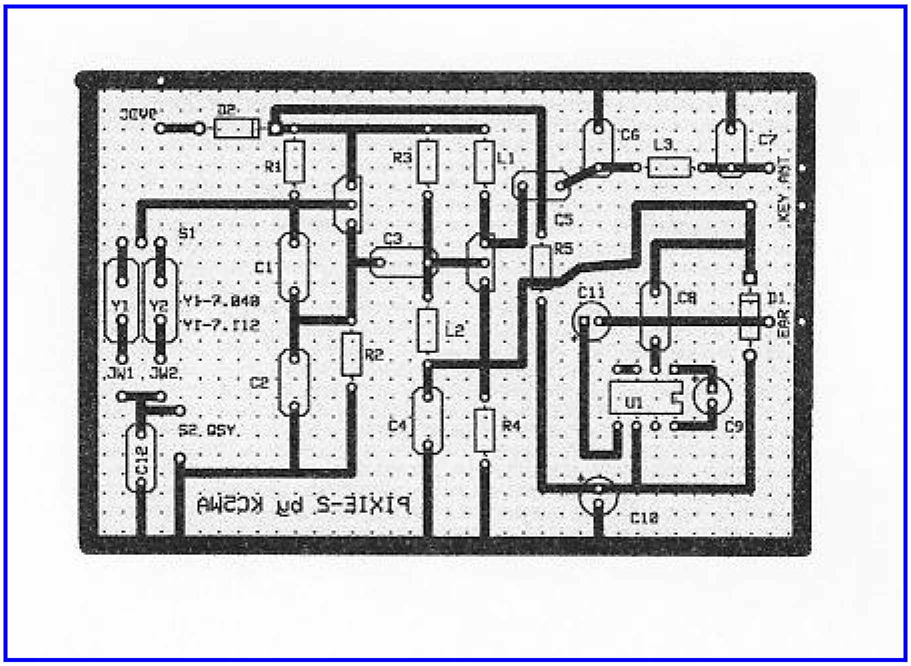
Color layout from Express PCB



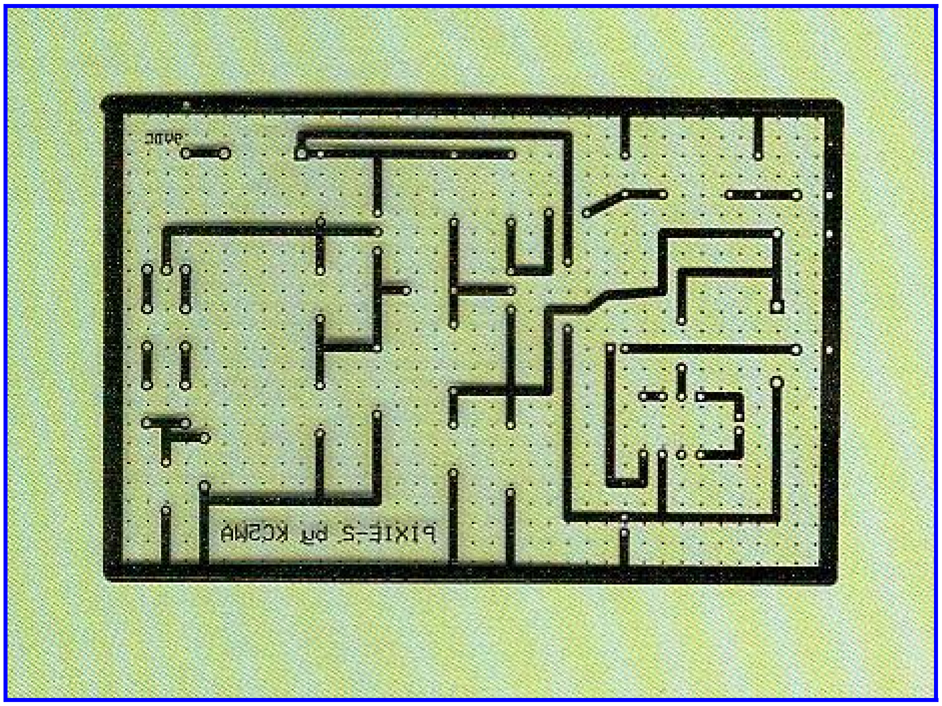
Etching layout



Parts Placement Silkscreen



Etching layout and Silkscreen combined



This is the Etching Transparency

I used a green sheet of paper behind the etching transparency to be able to tell the difference from the etching layout.

Parts List for my Pixie 2

*** Note The parts designations X1 and Y1, X2 and Y2 are interchangeable and are the same parts, "Crystals" because I GOOFED.

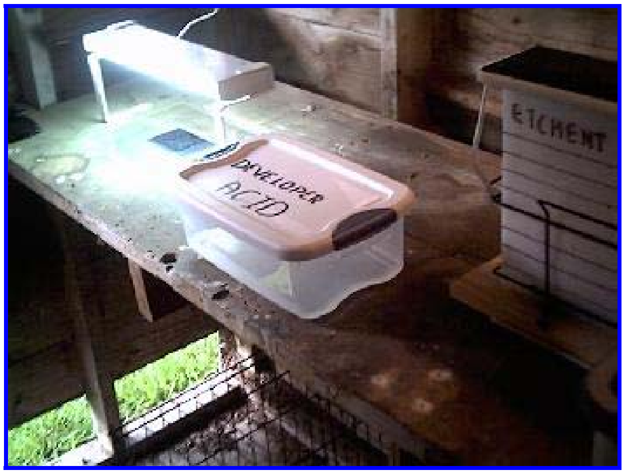
~~~~~~

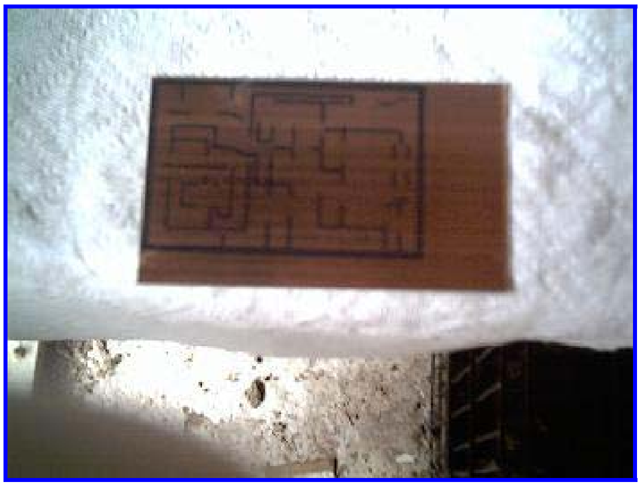
Here is my layout, an old work bench in the garage

~~~~~~

Etchant and rinse/waste water. Note when I mixed the sodium persulphate I had 4 liters of water after adding the 1kg of crystals I had 5 liters of solution.

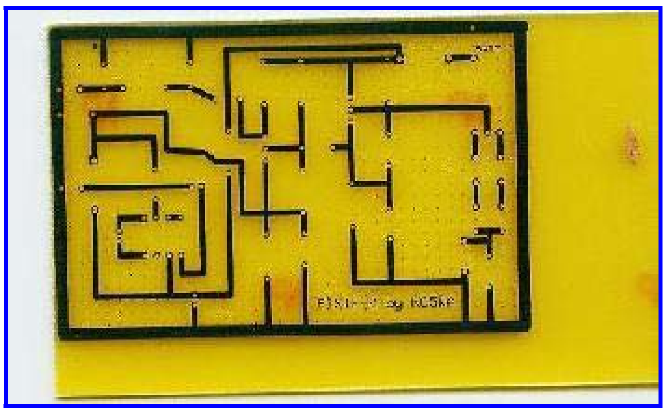
Exposure (10 minutes) of the photo resist pcb



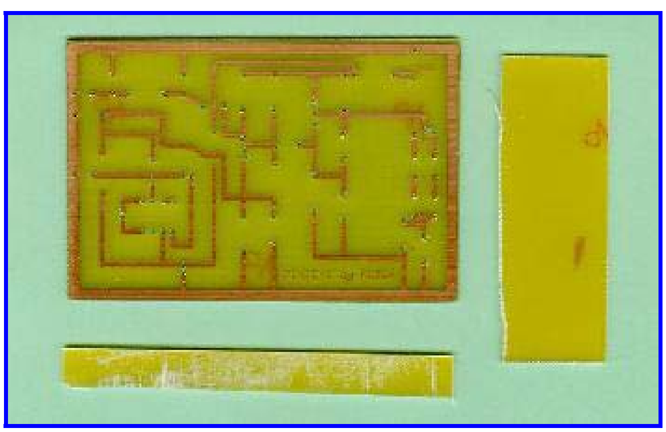


After exposure and the developer bath, even though the pictures are dark you can see the traces

Suspending the pcb in the etchant solution. I used some plastic clothes pins to accomplish this.

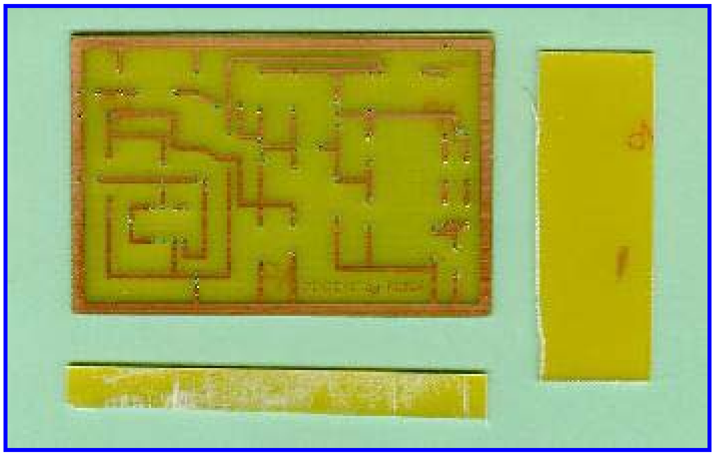


Using a hacksaw and 4" desktop work vice I trimmed off the excess board. One side of the circuit board was removed using the Dremel rotary cutting bit because the board would not fit in the vice. Then again using a Battery powered Dremel tool and the smallest bit I have on hand the holes were drilled. Then the board was cleaned using Acetone (finger nail polish remover) which also removed the last of the photo resist



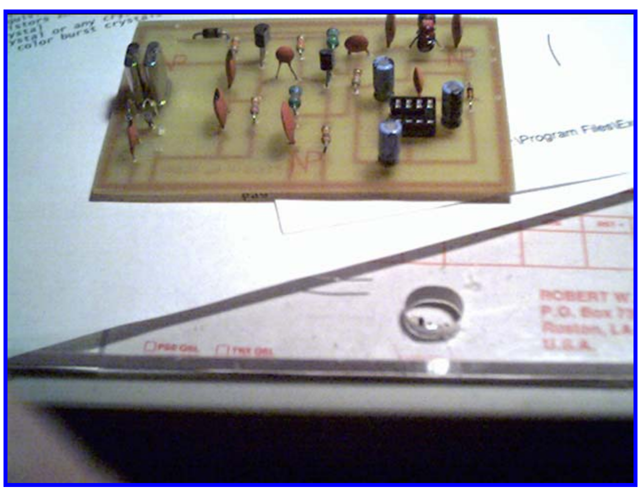
Here is the finished Pixie 2 PCB

~~~~~~



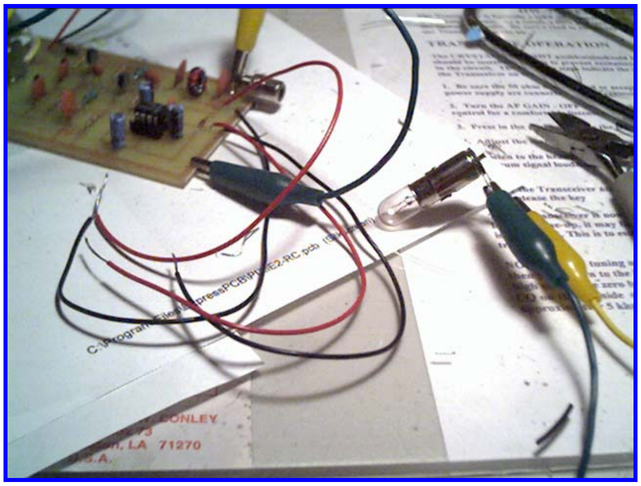
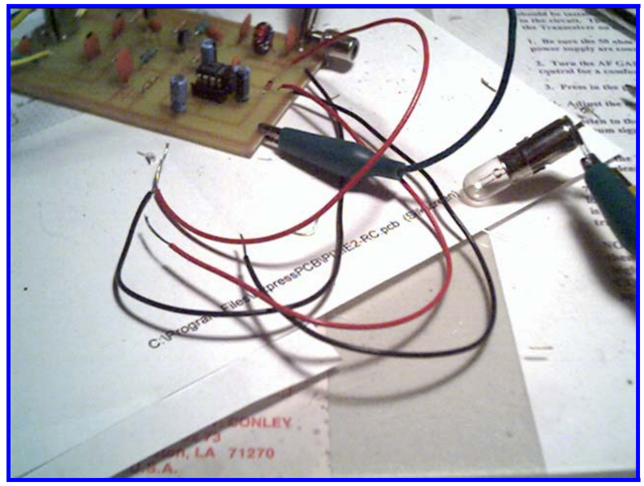
Here is the finished Pixie 2 PCB that was made earlier

Construction started

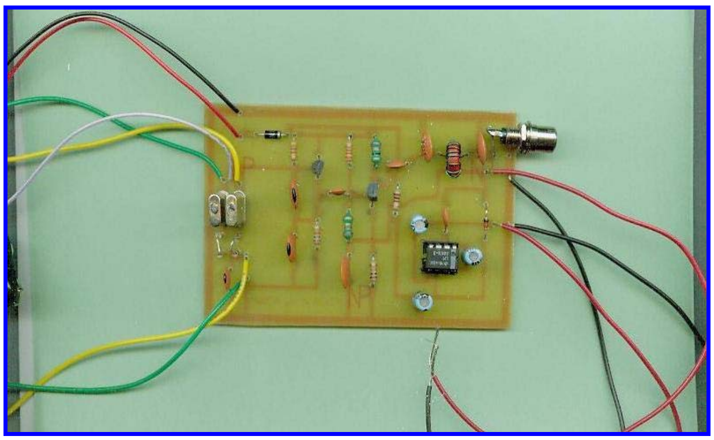


Almost finished, controls need to be added.

~~~~~~

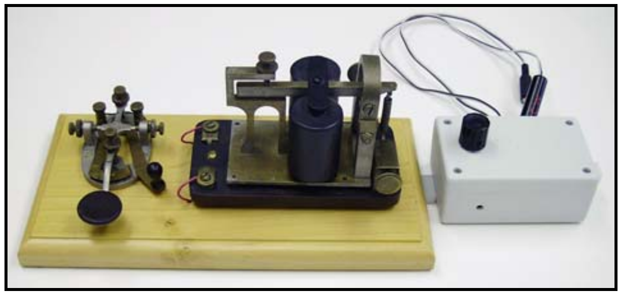
Controls added, an SPDT switch for selecting frequency. Either 7.040mhz or 7.122mhz notice the jumpers just below the crystals, inductor or capacitors can be added to "pull" the frequency. An SPST that puts a 68 pf capacitor in series to give a 700 khz offset for rx/tx



On the right a RCA phono plug for antenna, a pair of wires for the key and a pair for the ear plug.

Robert "RC" Conley, KC5WA may be reached by mail at 303 Riverbend Drive, West Monroe, Louisiana 71292, or by email at kc5wa-qrp@juno.com.

Mark Spencer (WA8SME)



"Slap, Click, Twang"

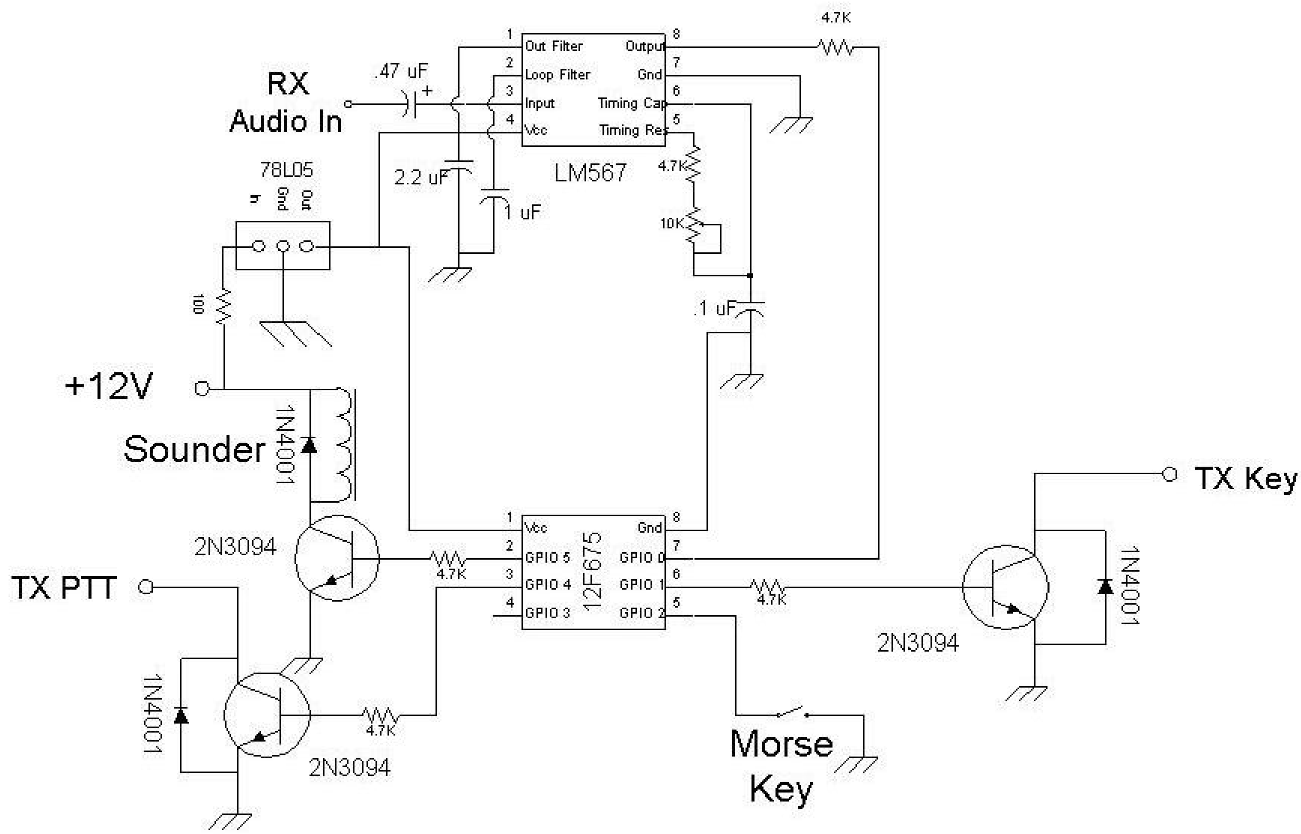
... Morse Code the Old Way

A simple interface that will allow you to use a telegraph sounder to copy code like they did in yester-year!

I have always wanted to do it. When I was a lot younger, there was a group of hams that used old telegraph sounders and American Morse Code on the bands. Their fists were meticulous; I admired their skill, more so because I couldn't quite achieve 100 percent copy while reading the mail. Later, I purchased at a ham fest a late 1880's vintage railroad telegraph sounder that I believe came from a New Hampshire railroad station. It has hung on my shack wall for years as a conversation piece and I use it periodically as part of a history lesson for the ham licensing classes I teach. But now the key and sounder have a new life and they bring a little nostalgic operation to the shack. This article describes a simple interface that will allow you to use a telegraph sounder to copy code like they did in yester-year.

The Interface Circuit

The main job of the sounder interface is to convert the CW audio tones produced by the receiver into electrical pulses that can drive the electromagnet coils of the sounder. An inexpensive integrated circuit, the LM567 tone decoder, makes the conversion a snap. The supporting components surrounding the LM567 in the circuit diagram will allow the IC to decode tones within the usual frequency range for CW work. The bandwidth of the decoder is about 100 HZ which helps reduce interference from adjacent QSOs. When a Morse dit or dah is received, the output of the LM567 goes to the low state (ground). It's that simple.



The second major component of the sounder interface is a 12F675 PIC microcontroller. Many will probably view the use of a PIC in the interface as overkill, but I prefer to view it as "old meets new." The PIC certainly simplifies the circuitry of the interface, reduces costs, and the software makes the interface easily adaptable to your particular situation.

Sounder Interface Software

The PIC software contains a loop that monitors the tone decoder for a low state (received dit or dah). When the low is detected, the PIC turns on the switching transistor that in turn energizes the sounder coils from a 12 volt DC power source. When the tone stops, the tone decoder goes to a high state and the PIC turns off the current to the sounder coils. A "slap, click, twang" sound is the result. The diode across the sounder coils (and across the other switching transistors) performs an important protection function. Very high, and potentially damaging reverse voltages can be produced within the sounder coils when the induced magnetic field collapses as the current is turned off. The diode provides a short circuit path for these reverse high voltages.

The PIC software also performs other interface functions. Closing the hand-key triggers an interrupt program that captures the attention of the PIC. The PIC follows your hand-key movement and keys the sounder, the PTT line, and the key line of the transmitter to allow you to produce not only code on the sounder, but also over the air. The PIC is programmed to hold the PTT line closed for a period of approximately 750 mS after the final character is sent

(VOX). Then the PIC releases the PTT line and begins to monitor for the next incoming signal from the tone decoder. The software source code is well documented and easily modified by the user. Contact the author for the source code.

Operating the Sounder Interface

Adjusting the sounder interface is simple. Tune in a CW signal as you normal would. Feed the audio to the interface and adjust the variable resistor until the sounder is activated in step with the incoming code. Continue to adjust the variable resistor until the sounder ceases to operate. Back the variable resistor adjustment to the half-way-point to center the tone decoder pass band to the audio frequency you normally use to copy code.

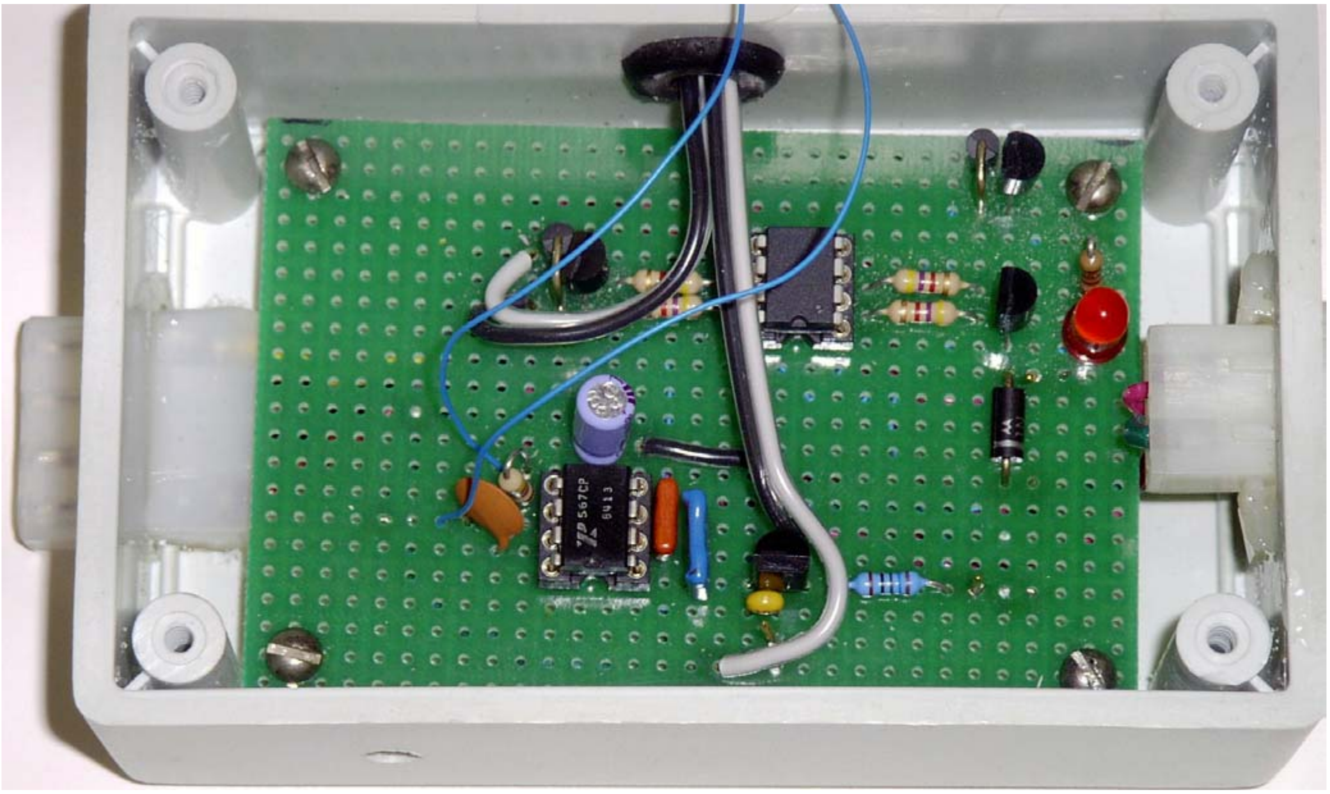
Operations with the Sounder

Operating with a sounder is something you have to experience for yourself. It was personally exhilarating the first time I connected up the interface, turned in W1AW code practice, and the sounder began its “slap, click, twang.” Even more surprising is that it didn’t take long before the “snap, click, twang” began to make sense. The transition from tone Morse Code to sounder code was easier than I thought it would be.

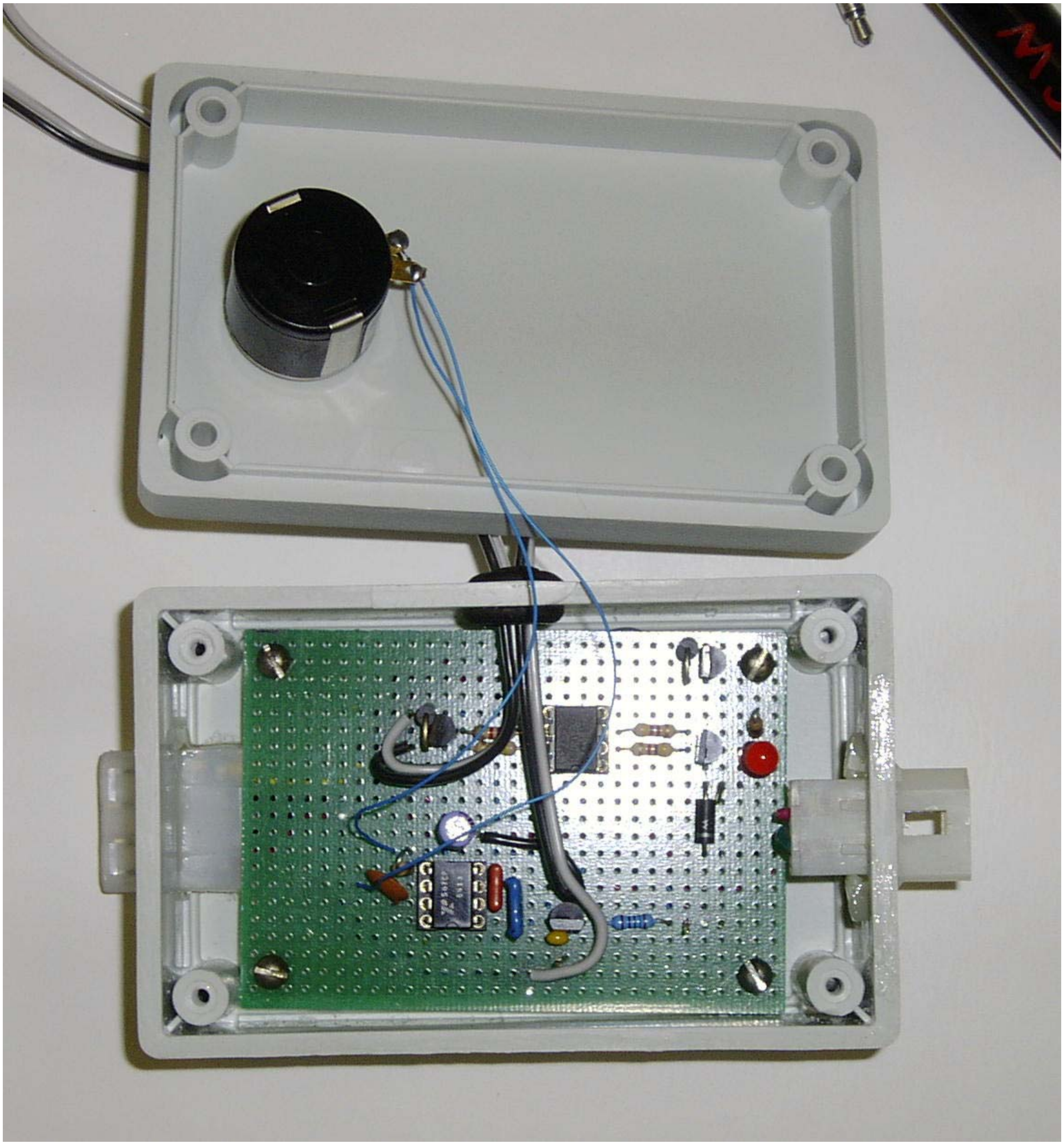
I’ve used the words “slap, click, twang” to describe the sound made by the sounder. The “slap” sound comes from the downward movement of the sounder arm as the energized coils pull it down hard against the sounder anvil. The “click” sound comes from the upward movement of the arm after it is released and the return spring tension pulls the arm up against a mechanical stop. The “twang” sound comes from the flexing spring. The dits and dahs make a different and distinctive “slap, click, twang” sound.

My first sounder QSOs were with K4JIF, K1TG, and WB0NQM. The narrow bandwidth of the LM567 tone decoder requires careful receiver tuning. The signals have to be relatively strong to maintain the rhythm of the sounder. Also speaking of rhythm, (though it could also be due to my inexperience) the sounder is not as forgiving of sloppy fists as normal tone CW. Finally, to prove to myself I could use a sounder, I did a W1AW CW proficiency run. With renewed pride (and probably not so humble) I copied 15 WPM.

So as the din of noise over the relevance and demise of Morse Code as a licensing requirement continues, why not step back and experience the code the way it was done in the beginning. A little “slap, click, twang” can perhaps put the din in a new perspective.



The Sounder PCB



The Sounder Internals

Mark Spencer, WA8SME
Project Coordinator,
ARRL Amateur Radio Education & Technology Program
ARRL Headquarters
225 Main Street
Newington, CT 06111-1494
(860) 594-0396
wa8sme@arrl.org
www.arrl.org/FandES/tbp
www.arrl.org/FandES/ead/teacher

AmQRP.ORG HOMEBREWER

Chuck Adams, K7QO

SPICE Modeling

... a Tutorial

SPICE, an acronym for Simulation Program with Integrated Circuit Emphasis, is a computer program developed at the University of California at Berkeley . Since it was developed with public funding, the program became public domain and is readily available from several sources over the Internet. I will give you pointers to sources that I know of at the end of this tutorial and hopefully you will not procrastinate before they disappear.

Keep in mind that SPICE was first developed on mainframe computers in FORTRAN in the early '70s. At that time in the computer revolution there was very little hardware available for doing graphics and SPICE may seem difficult to work with at some times due to graphics features being a later add-on. A number of individuals and companies have modified and extended the user interfaces to include graphics. There are even some frontend schematic programs (schematic capture) that produce data to be input to SPICE, but I have not done much research on these at this time

I have my own schematic routines that I used to produce all the schematic diagrams shown here along with the plotting software to graph the output. These are programs that I wrote from scratch in C and have been using for a number of years. They work and they do what I want them to do to get work done.

The early versions of SPICE up to the version called SPICE2G6 were done in FORTRAN, and I'll give you a pointer to the source. Starting with SPICE3 the program was rewritten in C, and the two current version numbers are SPICE3F4 and SPICE3F5. SPICE3F5 is just the previous version with bug fixes and some minor optimizations. You can also get the sources to these programs over the Internet.

Unless you have a lot of time on your hands, lots of programming experience, know a lot of electrical engineering and solid state physics, and you want to give up a lot of time on the air operating, I do not recommend that you spend much time working on the source files. There are more valuable ways for you to spend your time.

You do need the source and a compiler to get the program working on a large mainframe (if you have access to such a thing) or a non-Intel computer. If you have a PC running Windows95 or Windows98, then there is an excellent version of SPICE called WinSpice3 available for downloading off the Internet. I seriously recommend that you get it and run it. Several reasons for this. It is free and your costs will only be those incurred in downloading it if you are paying for connect time to your ISP. It is also not ``crippled'' in anyway, i.e. you may simulate large circuits with the program and it is not limited by parts or node counts like some demo versions of commercial software are.

Demo versions of commercial versions of SPICE are limited in what they can do and they give you only a few models of devices, and in some cases give you the wrong models and fail to bring this to your attention. The commercial version of SPICE and its variants is relatively expensive to upgrade to a full working version if you obtain the demo version. These programs are developed for commercial customers whose demands for support and more features require the additional costs to be passed on to the customer base. The commercial versions of SPICE will cost more than a brand new Yaesu FT-1000MP rig.

You are most likely the typical QRPer who experiments and builds things from scratch and you don't have an advanced degree in electrical engineering or do electronic design for a living. You probably research the ARRL Handbook or other literature and study circuits and borrow or use circuit designs for part of the work that you are doing. You probably find one or more circuits that provide the same function, read the specs written up by the designer or author and pick the one that you think best suits your needs. The criteria for selection may be price of components, access or availability of parts, complexity of the circuit, power consumption, etc. After the choice is made, you build the circuit and probably tinker with parts values to see if you can improve on the design in some way and make it better for your use.

Sometimes we succeed in this process and sometimes we fail due to any number of reasons. The problem is that failures, even though we learn more from them than we do our successes, may require a lot of time and sometimes money when we totally destroy expensive components. The use of a computer program like SPICE may help us to eliminate circuits that do not work or optimize those circuits that will work before we sit down with a prototyping board or do ugly construction. It is the purpose of this tutorial to show from the ground up some of the capabilities of the program and where I have found it to be useful.

Because of the wide range of experiences and education of the audience, I will assume no previous experience. For those that are experts already I hope to show you something new or unique along the way, but no guarantees are implied.

And another use for SPICE that I'm interested in is getting new radio amateurs and new individuals interested in electronic circuits, how they operate and learning how to use SPICE to analyze circuits and devices as they encounter them. SPICE makes a great teaching tool and even makes it more fun that sitting down with pen and paper and calculator and just cranking out numbers. A large number of engineering schools use SPICE in a similar manner.

WHY SPICE?

SPICE was developed during a period of transition in the semiconductor industry. In the mid-1970s the semiconductor complexity of chips and microcomputers was rapidly increasing. Prior to the production of an integrated circuit chip a prototype was made up of discrete components, tested and tuned, and then made into semiconductor material. The production of a single chip for testing and development was too expensive at the time and required long turnaround times for the manufacture of just a few parts to test and evaluate. And the testing and probing of the completed part was almost impossible in many cases due to the lack of test equipment and procedures at the time.

I remember seeing in an early issue of BYTE Magazine (around 1976 -- 1978 timeframe) photographs of the development team at Motorola in Austin, TX showing the prototype made up of discrete components for the Motorola 6809 microprocessor. This was done to develop the microprocessor chip, debug the logic, etc. Probing and measuring the logic states and transitions could only

be done on large systems at the time. But with the larger 68K series of microprocessors and everything since then, the only way to do microcomputer chip development has been with software such as SPICE and other programs. The scarey thing about the current state of computer development is that so much software and automation is needed that much of the detail is not known to humans but contained and embedded in the software development tools.

The side effect of this is that we as radio amateurs have access to a program that we can use for RF design, and we don't have to write programs from scratch.

INPUT TO SPICE

Remembering that SPICE was first developed in the days of computer input via card decks, you will find that the input is typically one 80 column line per component with the capability to continue long lines onto the next one. Some programs still limit the length of input lines to 80 columns. Also, case sensitivity occurs aperiodically, and you will know when this happens to you. I will show all source files for SPICE in upper case and in a typewriter font, and if I show lower case in the source it may be required for that instance. Sometimes the program does not give any warning messages or errors and just fails to output or do what we want. It is not an error on your part or the program. It is just the way it works, and you will learn rather rapidly where these things are critical.

First of all, let me to list the types of components that SPICE can simulate and the letter used for each.

C Capacitor

D Diode

E Voltage-Controlled Voltage Source

F Current-Controlled Current Source

G Voltage-Controlled Current Source

H Current-Controlled Voltage Source

I Independent Current Source

J Junction Field-Effect Transistor (JFET)

K Coefficient of Coupling for Mutual Inductance

L Inductor

M Metal-Oxide Semiconductor Field- Effect Transistor (MOSFET)

Q Bipolar Junction Transistor (BJT)

R Resistor

T Transmission Line

V Independent Voltage Source

I will give an example of most of these, but due to limited time and space will be unable to show you each and every use for typical radio amateur applications.

First, we need to define a ``node''. A node is a point where two or more components are physically connected together. Think of this as a solder joint or two or more pads on a circuit board that are connected together via the same trace on the board. Each node in a SPICE circuit is given a number and in some programs you can use names, but I will try to restrict myself to numbers in this tutorial to avoid confusion and hopefully avoid making things more difficult. The numbers are positive integers or zero, but zero is reserved for the ground reference point. Be sure to always assign the ground connected points or nodes the value zero. All other node voltages are referenced with respect to this point in the circuit.

The node numbers are arbitrary. You just take the schematic of the circuit that you wish to simulate and assign unique numbers to each of the nodes in the circuit in much the same way that you assign numbers to the resistors and capacitors and other parts. Remember that the only restriction is that 0 is reserved for ground. The program does not care if you skip numbers, and it will warn you if you have a component with one node that is not connected to another part of the circuit. I would suggest that you incrementally assign node numbers and be extra careful about not duplicating a number between two different nodes. You'll in effect connect the two points together with the program giving you no warning that an error on your part has occurred.

For each component you assign a name, the node value(s), and the component value. There also may be options that you can add for temperature coefficients or initial conditions for the start of the simulation. I will try to work in an example before I run out of time and space.

Since the component values may have a wide range of magnitudes, SPICE will allow you to use factors for component values from the following choices and the letter does not need to be uppercase, but I tend to use uppercase except for micro,

F = 1E-15 (femto)

P = 1E-12 (pico)

N = 1E-9 (nano)

U = 1E-6 (micro)

M = 1E-3 (milli)

K = 1E3 (kilo)

MEG = 1E6 (mega)

G = 1E9 (giga)

T = 1E12 (tera)

where E denotes the power of 10, so that 1E-15 is 10^{-15}. Please be very careful. The abbreviations for the multipliers or scaling factors will come back to haunt you. Make sure that if you want a one megohm resistor that you use 1MEG or 1E6 and not 1M, as the latter will give you a thousandth of a ohm and you could have just as well put in a short or zero ohm resistor. Also, the letter F should not be used in capacitors unless preceded by one of the other multipliers. For example, 0.01UF or 0.01uF would be a correct notation for the value of a 0.01 microFarad capacitor but 0.1E-6F would be a capacitor with a value of 10^{-21} Farad.

This one error alone has cost some companies and consultants a lot of time and money. Fortunately I was not the one involved.

If the value consists of a string of numbers and then letters, the first letter is taken as a scaling factor (if it is one of the above) and the rest are ignored except for the case of MEG as a multiplier. This allows one to use uF for microFarad and 1MEGohm to denote the obvious one meg value for a resistor or 50ohms for a 50 ohm resistor since o is not a valid scale factor or multiplier.

RESISTORS

A resistor is placed in the circuit between nodes N1and N2 with the line

Rname N1 N2 value

where name is the name of the resistor, usually in the form of a number, and the resistor has the numerical resistance of value ohms. For example

R36 36 37 1.2K

represents a 1.2K resistor connected between nodes 36 and 37.

I have shown the lines above and will continue to show the input lines for SPICE in this tutorial indented some to offset them from the text. The lines of input to a SPICE program must begin in column 1; thus the R in the above lines must be in column 1.

There is an optional field on the resistor input line for temperature coefficients if one wants to do temperature analysis of a circuit, but I prefer to leave that for more advanced work in another place and time. Just remember that the fields exist for the input and you will have to research a complete book on SPICE to get the details if you feel the need to use it. It is critical for semiconductor chip development.

CAPACITORS

A capacitor is placed in the circuit between nodes N1 and N2 with the line

Cname N1 N2 value

where name is the name of the capacitor, usually in the form of a number, and the capacitor has the numerical capacitance of value Farads. For example

C56 103 0 0.047uF

represents a 0.047(F cap connected between nodes 103 and 0 (ground).

In some books you will see authors use the notation

Cname N+ N- value

to represent the input line for a capacitor. The notation N+ for the positive node and N- for the negative node for non-electrolytic caps. SPICE couldn't care less about this, and don't let it get to you. A capacitor is a capacitor, and only in the real world do we get into trouble by hooking one up backwards like a tantulum or electrolytic. SPICE will also allow one to specify a non-linear capacitor with polynomical coefficents that determine the value of the capacitor dependent upon the voltage across the terminals. I assume that we will not be interested in this capability for now. This is useful for working with varactors if you have the curve for the voltage vs. capacitance characteristics. I'll try to come up with a variable bandwidth crystal filter using this as an additional circuit to try, either here on the web page at a later date. Remind me if I haven't done it by the end of April 1999.

INDUCTORS

For an inductor the input line consists of

Lname N+ N- value

for a value Henry inductor connected between the two nodes N+ and N-. The notation is such that a positive increasing current flowing from the positive node to the negative node will induce a positive voltage across the inductor with N+ being the positive or higher potential. So an input line in the form of

L3 27 39 15uH

would represent L3, a 15(H inductor, connected between nodes 27 and 39.

One thing that might help keep you out of trouble is to use the ``dot" notation for inductors and transformers and always use the N+ node for the "dotted" end of the inductors in the circuit. I'll illustrate this in the next section on MUTUAL INDUCTORS.

MUTUAL INDUCTORS OR TRANSFORMER

The line of the form

Kname Lname1 Lname2 value

allows two inductors, Lname1 and Lname2, described elsewhere in the source file for the circuit, to be magnetically linked by a coupling factor value. This is nothing more than a transformer. The value must be greater than zero and less than or equal to one.

K1 L3 L4 1.0

would represent a tightly coupled transformer with L3 and L4 being the two windings and the coupling factor being 1.0. The N+ nodes of the two inductors would be the dotted ends during the SPICE calculations.

There is no limit to the number of windings that can be magnetically coupled, so you can work with trifilar windings if you want. Say we have N windings on the same toroid. We would need N*(N-1)/2 coupling factors to describe this transformer. The way you can determine this is that for every winding there are N-1 remaining windings that we can interact with, but the coupling is the same between Li and Lj and Lj and Li, thus the division by two.

Let's look at a transformer with two windings on a toroid. Let's also look for the ease of getting into what is going on, say that the core is a T37-6 with 8 turns for the primary L15 and 16 turns for the secondary L16. Knowing the core and the number of turns gives us 0.192(H for L15 and 0.768(H for L16.

The SPICE source lines for this transformer would consist of the three lines

L15 55 56 0.192uH

L16 57 58 0.768uH

K1516 L15 L16 0.995

I have not seen in the literature factors less than 0.90 and have I have not had a chance to do the experiments myself to see just what values may be used for the coupling factor to get exact results. I would be glad to hear from those who have done research in this area and it may be an area for someone to do an article in for the QRP ARCI Quarterly or other newsletter(s). Let me know if you have or are planning to do this as it may save me some duplication of effort. If the simulation gives you the same results as you get from the real circuit, then you have succeeded in doing it correctly.

The transformer shown in the above three lines would be represented by the following schematic. If you use a single notation for the transformer, then you can call the transformer T1 and then in the SPICE input file put a LT1P inductor for the primary and a LT1S inductor for the secondary to make the notation easy to follow.

Figure 1. Transformer in SPICE.

If I forget to do it manually, go ahead and place two dots for the two upper terminals in the transformer above as the schematic routine doesn't have it in there yet --- work in progress, as I always say.

Thanks.

INDEPENDENT VOLTAGE SOURCES

Independent voltage source is a term that refers to voltage sources within a SPICE simulation that are not determined or influenced by outside conditions or other circuit parameters. If you have a 12.68V supply, then you will have 12.68V across the terminals of the supply whether the current through it is 1(A, 1mA or 1,000A, difficult to do in real life but very easy for the computer to do. We are emulating an ideal battery or other ideal power source for the power sources.

For independent voltage sources the SPICE input line may have several formats, so let me take a few examples to illustrate how we provide voltage sources.

First, the generalized format is

```
Vname N+ N- [ [[DC DC-VALUE]
     [AC [ACMAG [ACPHASE]]]
     [TRANKIND(TPAR1 TPAR2 ...)] ]
```

Now don't panic. I know it looks complicated. The brackets [and] indicate the beginning and ending of optional fields.

Let's first take just a simple DC 12.8V supply. This would be represented by

```
Vcc 31 0 DC 12.8V
```

and please note that here I used letters for the name instead of numbers. Some versions of SPICE may require you to use numbers and not allow alphabetic characters in the field name, so you may have to experiment if you are not using SPICE3F4 or WinSPICE3. One thing that may happen or you will want to know about is what happens if you happen to leave out the DC and voltage value. You will get a voltage source with a DC value of zero in such instances. The voltage acts like a short and provides no energy to the circuit. You can use this voltage source as an ammeter. SPICE does not have a formal definition of an ammeter but with a voltage source of zero you can print or plot the current through the voltage source and we will make use of this later, so don't forget about it.

```
Vammeter 31 0  DC  0V
```

would be a typical line in a SPICE simulation to indicate that the voltage source is going to be used as an ammeter to measure current through that part of a circuit.

OK, so much for DC, let's look at constant magnitude AC voltage sources. These we can represent with the source line

```
Vac 31 0  AC 1.50V
```

to represent an AC voltage source of 1.5 volts. The voltage will be constant independent of the current through it and the load on it from the rest of the circuit, sometimes impossible to do in real life. Note that nothing is said in this line about the frequency, and we need another control line in the SPICE source file to set the frequency. We'll get to that later, or we can use one of the transient analysis parameters shown in the next few paragraphs.

INDEPENDENT CURRENT SOURCES

Like the independent voltage sources, the generalized input format for independent current sources is

Iname N+ N- [[[DC DC-VALUE]

[AC [ACMAG [ACPHASE]]]

[TRANKIND(TPAR1 TPAR2 ...)]]

where a positive current flows from the positive terminal through the source to the negative terminal. Instead of having a voltage source we have a current source and can do the same thing for currents that we can do for voltages. All we have to do is substitute I for the V and use the same control structures that we will see for the voltages when doing transient analysis.

FIRST CIRCUIT ANALYSIS USING SPICE

Since we are now far along with a description of SPICE and working on what it can do, let's take a little side trip and do an actual simulation using SPICE3 both on a workstation and on a PC. I'll be using SPICE3F4 on the workstation and WinSpice3 on the PC running Windows98 in these examples. Hopefully you have one or the other to work with.

In order to get you on the computer early, let's start with a simple passive circuit that only has resistors, capacitors, and inductors and no active devices like transistors or diodes. We'll start with the Five Pole Chebyshev Low Pass Filter that is typically found as the final filter in the PA section of a QRP transmitter. If you have an ARRL Handbook for 1995 or later, go to Chapter 30 page 23. This is 30.23 using the current page numbering scheme.

The schematic for this filter looks like:

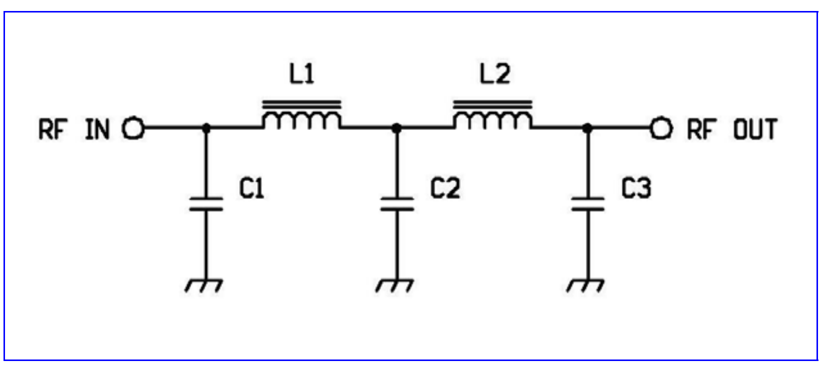


Figure 2. Generic Five Pole Chebyshev Filter Schematic.

The output and input design impedances for this filter are 50 ohms and in order to correctly simulate its behaviour we must make sure to setup 50 ohms for the source impedance and terminate the output to a 50 ohm load. This means that we now have a circuit that looks like:

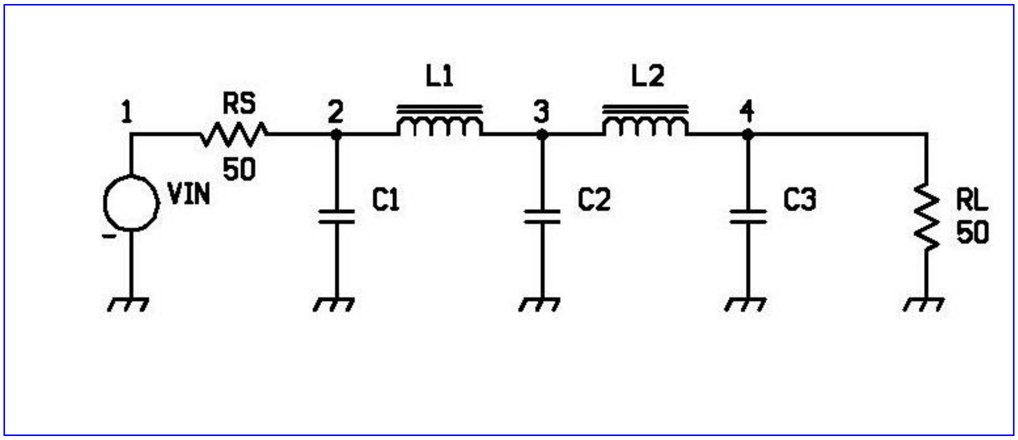


Figure 3. Chebyshev SPICE Simulation Circuit.

We have the transmitter section represented with an internal impedance of 50 ohms and a voltage generator represented by VIN and RS and the load resistance represented by RL, also 50 ohms. I have assigned node numbers to each of the nodes other than ground which will always be node 0, so I won't bother to show it in any diagram that you see from me. Saves me time and energy.

SPICE typically expects to have the input source for simulation to be a file with a name and extension in the form of FILENAME.CIR, where FILENAME is some name that you chose. So what I'll do for this little exercise is give the file a name of CHEBY01.CIR. Every input file for SPICE has the first line treated as a comment or title line, so make sure that the first line is not supposed to be one of the elements in the circuit. I usually put the name of the file that contains the source and some other comments that remind me just exactly what I was doing. If you need to put comments in the source file, then start each comment line with a * or asterisk in column 1. Any line that you see with a plus sign + in column 1 is a continuation of the previous line. This allows us to have source statements that are more than one line long if we need them and we will.

The source file must end with a line that has .END on it. Some of the newer versions of SPICE may allow additional statements after the .END, but they are typically some control statements.

Also, to make things more complicated we need to tell SPICE to output some information for us either as text or to plot some data that we are interested in. Any commands that are bookkeeping or commands to SPICE to do something all start with a period in column 1 and the command name immediately following it. Some of the things that we can do are PRINT values or PLOT values, and I'll explain these as we go in each of the samples. I will come back and give you the general syntax of the commands after we get some initial runs made and then move on to more complex circuits.

OK, with that all said and done, let's try a simulation of the Five Pole Chebyshev Low Pass Filter. The file will have the name CHEBY01.CIR where the 01 in the name I use to indicate that this is file number one of a possible series of runs involving Chebyshev filters. The file for Figure 3 will look like:

```
CHEBY FILTER  chebyo1.cir
*
*   FIVE ELEMENT CHEBY FILTER
*   NR. 91 from ARRL Handbook
*
*   K7QO sample file for SPICE
*   introduction
*
VIN 1    0      AC 1
RS     1 2      50ohms
C1     2 0      390pF
L1  2 3 1.48uH
C2     3 0      750pF
L2  3 4 1.48uH
C3     4 0      390pF
R1  4 0 50ohms
*
*   The following line tells SPICE what
*   range of frequencies to use for the
*   input source VIN
*
.AC LIN 81 6MEG 21MEG
*
*   The following lines tell SPICE2G6 to *       print values and to plot values.
```



```
*   I use the PRINT command to output *        results to a file in order to plot the
*   results in my own format instead of
*   the format that will appear on a
*   screen for graphically based SPICE
*   versions.
*
.PRINT AC VDB(1) VDB(4)
.PLOT  AC VDB(1) VDB(4)
*
.END
*
*   The following line used with
*   WinSpice3 to get automatic
*   execution of the program and
*   plotting.
*   The * in column 1 is necessary.
*
*   #plot VDB(4)
```

TRANSIENT ANALYSIS

For both independent voltage and current sources we have a parameter, TRANKIND, that allows us to set the type of transient analysis. There are five available types of transient analysis values allowed in SPICE simulations and they are

- **PULSE --- for a single pulse or pulse train**
- **SIN --- for a sine wave that may have a decay factor**
- **EXP --- for a single pulse with an exponential rise and fall**
- **PWL --- piece-wise linear function**
- **SFFM --- single-frequency frequency modulation source**

PULSE TIME DEPENDENT VOLTAGE SOURCE

The first of these is the pulse, which may be just a single pulse or a series of pulses. The source line for the voltage (the current time dependent sources are the same as the voltage except with an I) pulse is

Vname N+ N- PULSE(V1 V2 DELAY UP DOWN WIDTH PERIOD)

where

- **Vname --- the name for the voltage source**
- **N+ --- the positive terminal of the voltage source**
- **N- --- the negative terminal of the voltage source**
- **PULSE --- the type of transient analysis to be done**
- **V1 --- the starting voltage of the pulse in Volts**
- **V2 --- the pulsed voltage value in Volts**
- **DELAY --- the delay time before the start of the pulse train**
- **UP --- the rise time for the pulse in going from V1 to V2**
- **DOWN --- the fall time for the pulse in going from V2 back to V1**
- **WIDTH --- the time that the voltage remains at V2 during the pulse**
- **PERIOD --- the time interval between beginnings of each pulse if more than one**

where all the times are in seconds.

In order to show you exactly what such a pulse looks like and to give you a circuit to input into SPICE to examine the nature of these parameters, here is a test circuit using a single pulse.

```
TRANSIENT ANALYSIS of a Pulse Train
*
*  pulse01.cir   Chuck Adams, K7QO
*
*   Pulse is delayed 10mS
*   Starts at voltage level of 1.0V
*   Rise time of 1mS
```



```
*    Pulse Voltage of 2.5V

*    Pulse Width of 13mS

*    Fall time of 2mS

*    Period of 20mS

*

Vinput 1 0 PULSE(1.0V 2.5V 10mS 1mS 2mS 13mS 20mS)

Rload  1 0 1ohm

*

*  The .TRAN command to SPICE is

*  necessary in order to start up the

*  simulation with the pulsed voltage

*  source

*

.TRAN 0.1mS 75mS

*

.PLOT V(1)

.END
```

The plot for the output looks like:

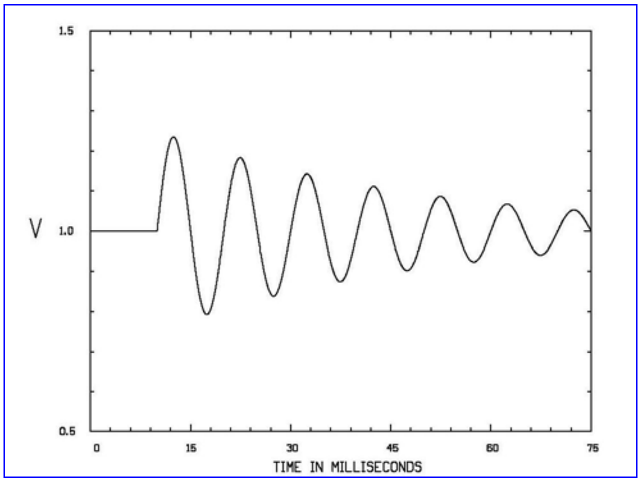


Figure 4. Output from transient analysis.

If you make the period value equal to zero, you will only get a single pulse. This feature is needed and useful to study oscillators as you need a condition to start oscillation. The reason for having the function for time dependent voltage or current sources is to also determine response functions for devices or functions for advanced studies and critical analysis in more demanding areas of circuit simulation.

SINE WAVE TIME DEPENDENT VOLTAGE SOURCE

Another wave form for a voltage source is the sine wave. Its input format is:

Vname N+ N- SIN(OFFSET

AMPLITUDE FREQ DELAY

DECAY)

where

- **Vname --- the name for the voltage source**
- **N+ --- the positive terminal of the voltage source**
- **N- --- the negative terminal of the voltage source**
- **SIN --- the voltage source is a sine wave**
- **OFFSET --- the offset voltage for the source**
- **AMPLITUDE --- the voltage amplitude for the sine component of the source**
- **FREQUENCY --- the frequency for the sine component of the source**
- **DELAY --- the time delay before the sine wave starts**
- **DECAY --- the exponential decay factor for the damped wave**

The way this works is that the voltage will start out at OFFSET volts until time is equal to DELAY. Then the sine wave will start up with an amplitude of AMPLITUDE and frequency of FREQUENCY. If DECAY is non-zero, then the amplitude of the sine wave will die off until the simulation stops.

Once again here is a sample circuit as above in the case of the pulse but now with a sine wave source superimposed upon a DC offset voltage. I'll go ahead and show the input file and the plot. I do hope that you will take the file and use it to vary the parameters and look at the results.

TRANSIENT ANALYSIS of a Sine Transient Function

*****

*** sine01.cir Chuck Adams, K7QO**

*****

***   Sine Wave of 100Hz with amplitude**

***   of 0.25V delayed 10mS with *damping factor of 25.0**

*****

Vinput 1 0 SIN(1.0V 0.25V 100Hz 10mS 25.0)

Rload 1 0 1ohm

*****

***. TRAN TIMESTEP TIMESTOP**

***   TSTART TMAX**

*****

***   If you run this simulation, because of**

***   the 100 sample points per mS and the *   long plot time of 75mS you will note * that this takes a few seconds to do**

***   before SPICE comes back. But I can *   tell you this is a lot faster than in the * old days of the mainframes.**

*****

***   I needed the extra fine resolution to**

***   produce nice curves in the plot. Play *   with this. You'll learn why I did it**

***   this way.**

*****

.TRAN 0.01mS 75mS 0 0.01mS

*****

.PLOT TRAN V(1)

.END

Please note the scales for the following plot of the output from the damped sine wave transient analysis. Because I used my own plot routines, I was able to scale to the factors that I wanted. When you run WinSpice3 it will automatically scale it for you and you may get a slightly different looking graph. The importance is that the information comes from the program in a graphical format and it becomes almost instantly recognizable for what it is, i.e. a dampled sine wave. Once again I recommend that you take the above simple program, run it on your own and experiment with the values. It is the only best way to help you remember the parameters and their meanings.

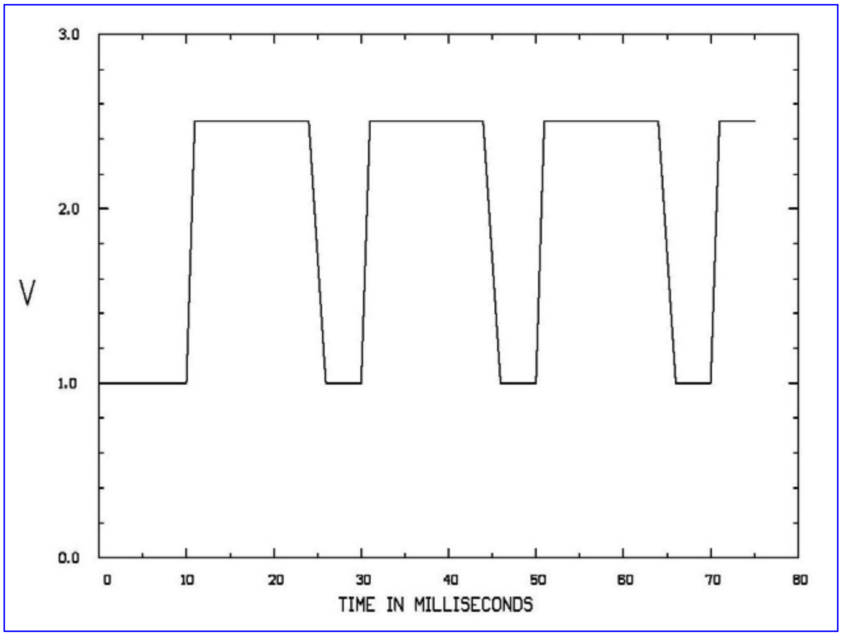


Figure 5. Output from sine wave transient analysis.

I am going to take the liberty of skipping the other forms of transient analysis for this tutorial due to restricted space. If there is enough demand for the information I would be glad to write it up at a later time. Again, there are books available that contain the details if you feel the need to pursue it further. The remaining forms are rarely used in amateur work.

DIODES

A diode in SPICE is modeled after all the physics of semiconductor materials for a PN junction. There are a number of parameters that go into the behavior of all the kinds of diodes that you can use. The difficulty is finding a model (the values of all the parameters needed) for the part number that you may have but fortunately the more popular diodes such as the 1N914 or 1N4148 silicon diodes are readily available. Models for the 1N34A and some others are more difficult to come by and may only exist in expensive libraries of models only available in the larger more expensive commercial versions of SPICE. I have spent considerable personal time surfing the Internet collecting all the models of diodes, transistors, and anything else that I could find to keep on hand for SPICE simultations.

In order to use a diode in SPICE you need two lines. The first is similar to the input lines for a resistor or other component used to specify the name and nodes but instead of specifying a value you point to a model of the diode. The line has the form

Dname N+ N- MODELNAME

[AREA] [OFF] [IC=VD]

where

- **Dname --- the name for the diode in the circuit**
- **N+ --- node number for the anode of the diode**
- **N- --- node number for the cathode of the diode**
- **MODELNAME --- the modelname used to setup all the parameters for the model representing the diode**
- **AREA --- an optional parameter of concern to silicon manufacturers on the area for the junctions inside a device**
- **OFF --- an initial condition for DC analysis of the circuit**
- **IC=VD --- the initial voltage across the diode for any transient analysis**

The general format for the model of a junction diode is

MODEL MODELNAME D(PAR1=PVAL1 PAR2=PVAL2 ...)

where MODELNAME is the name to be referenced in the line defining the diode in the circuit.

The parameter names, PAR1, etc. that are available and the parameters that they represent are from the following list of 14 possible references. You do not have to give them all when setting parameters for a diode. Some of the parameters have default values and hopefully you won't have to worry about this if you already can find a complete model for the diode that you are using.

- **IS --- saturation current**
- **RS --- ohmic resistance**
- **N --- emission coefficient**
- **TT --- transit time**
- **CJO --- zero-bias junction capacitance**
- **VJ --- junction potential**
- **M --- grading coefficient**
- **EG --- activation energy**
- **XTI --- saturation-current temperature exponent**

- **KF --- flicker noise coefficient**
- **FC --- coefficient for forward-bias depletion capacitance formula**
- **BV --- reverse breakdown voltage**
- **IBV --- current at breakdown voltage**

So, here are two samples of the use of diode models, one for the 1N34A or 1N270 Ge-diode and one for the 1N4148 or 1N914 Si-diode. These were created to check out the model with a real live diode setup in the following test circuit.

```
1N34A DIODE MODEL
*
*   1N34A.cir   Chuck Adams, K7QO
*
*   Test 1N34A Model Circuit
*   1N34A Ge Diode Model from Roy
*   Lewallen, W7EL
*
VIN 1 0
*
*   Voltage source setup to be used as
*   ammeter
*
VTST 1 2 DC 0
D1 2 0 1N34A
.PRINT DC V(2) I(VTST)
.PLOT  DC V(2) I(VTST)
*
*   Voltage from -1V to +0.5V in 0.01V
```



```
*   step
*
.DC VIN -1.0V 0.50V 0.01V
*
*   Diode Model for 1N34A
*
.MODEL 1N34A D(BV=75
+   CJO=0.5E-12 EG=0.67 IBV=18E-3
+   IS=2E-7 RS=7 N=1.3 VJ=0.1
+   M=0.17 )
.END
```

```
Diode Model Test of 1N4148
*
VIN 1 0 DC 1
VTST 1 2 DC 0
D1 2 0 D1N4148
.PRINT DC V(1) V(2) I(VTST)
.DC VIN -1.0V 1.00V 0.02V
.MODEL D1N4148 D(IS=0.1p RS=16 CJO=2p TT=12n
+   BV=100 IBV=0.1p)
.END
```

The following schematic was used to perform the experiment to measure the voltage-current curves for both the 1N34A and 1N4148 diodes and compare with the simulation results for the diode models. Care was used in accounting for current and voltage drops due to intrumentation and time was taken to avoid thermal junction effects where possible.

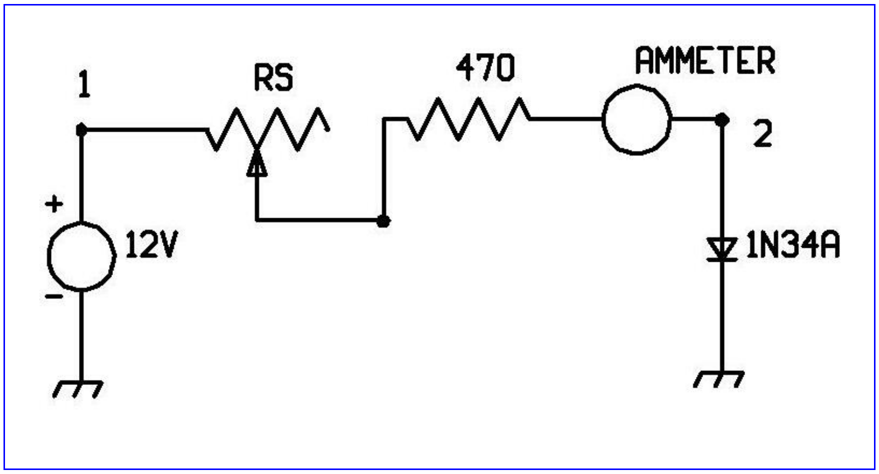


Figure 6. Diode test circuit.

Then the following graph was prepared from the output of the simulation runs and the measured data from the experiment. Excellent agreement was obtained, and variations may be attributed to variations in manufacturing processes. In order to further investigate the models versus actual physical measurements, a more detailed and traceable history of the part (some may have been manufacturing rejects) must be made and a greater number of samples measured. The purpose here is to illustrate the process of modeling and comparison to real physical results. Some parameters in the models may be changed carefully to match the measurements but with great care. The solid curves are from the SPICE model simulations, and the data points are from the physical measurements.

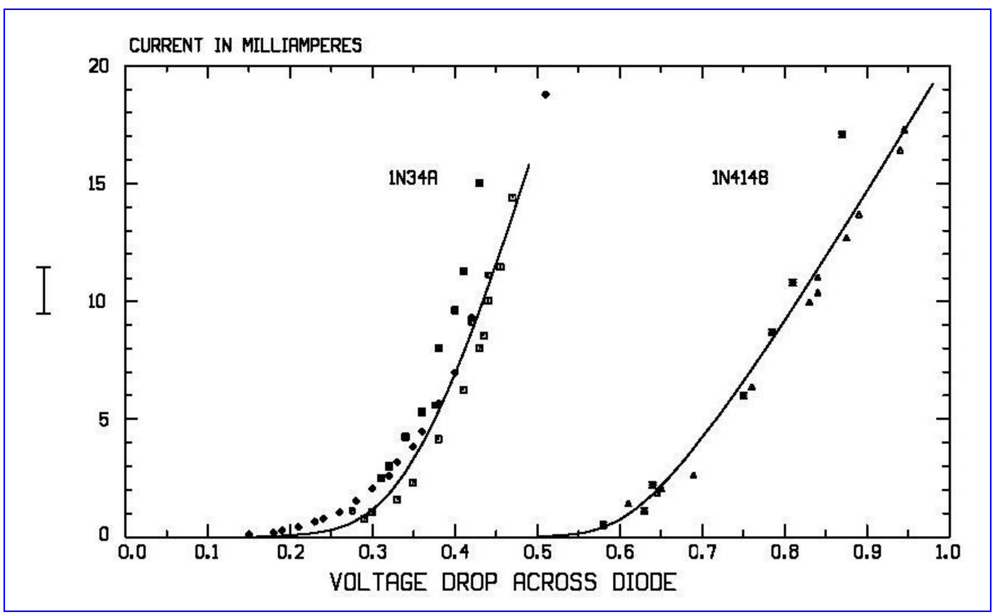


Figure 7. V-I characteristics for 1N34A and 1N4148 diodes.

You are now at the point whereby you can model simple circuits that include the diode. You can now model an RF probe, a half-wave rectifer, a full-wave rectifer, a voltage doubler, and various other circuits that involve the use of diodes and passive devices in their circuits. So take some time and look in the ARRL Handbook and see if there are some circuits that you can now simulate. It will be fun. Only with experimentation with the program can you really grow to appreciate its capabilities.

Bipolar Junction TRANSISTOR --- BJT

The next device to look at is the transistor in SPICE. The first one to consider is the BJT. The format for the device that we will use is

Qname NC NB NE MODELNAME

where the files are

- **NC --- node number to which the collector is connected**
- **NB --- node number to which the base is connected**
- **NE --- node number to which the emitter is connected**
- **MODELNAME --- model name for the transistor parameter definitions**

The BJT model line has the format similar to the diode model line, but has one for NPN transistors and one for PNP.

.MODEL MODELNAME

NPN(PAR1=PVAL1

PAR2=PVAL2 ...)

.MODEL MODELNAME

PNP(PAR1=PVAL1

PAR2=PVAL2 ...)

There are 40 model parameters that may be used to completely describe the transistor. I'll pass on listing them as I did for the diode. Let me just show you a sample of three commonly used transistors.

.model PN2222A NPN(Is=14.34f

+ Xti=3 Eg=1.11 Vaf=74.03 Bf=255.9 +Ne=1.307 Ise=14.34f Ikf=.2847

+ Xtb=1.5 Br=6.092 Nc=2 Isc=0

+ Ikr=0 Rc=1 Cjc=7.306p Mjc=.3416

+ Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 + Vje=.75 Tr=46.91n Tf=411.1p Itf=.6 + Vtf=1.7 Xtf=3 Rb=10)

*** Fairchild case=TO92**

.model 2N3904 NPN(Is=6.734f Xti=3 + Eg=1.11 Vaf=74.03 Bf=416.4

+ Ne=1.259 Ise=6.734f

+ Ikf=66.78m Xtb=1.5 Br=.7371 Nc=2 + Isc=0 Ikr=0 Rc=1 Cjc=3.638p

+ Mjc=.3085 Vjc=.75 Fc=.5

+ Cje=4.493p Mje=.2593 Vje=.75

+ Tr=239.5n Tf=301.2p Itf=.4 Vtf=4

+ Xtf=2 Rb=10)

*** Fairchild case=TO92**

.model 2N3906 PNP(Is=1.41f Xti=3

+ Eg=1.11 Vaf=18.7 Bf=180.7 Ne=1.5 + Ise=0 Ikf=80m Xtb=1.5 Br=4.977

+ Nc=2 Isc=0 Ikr=0 Rc=2.5

+ Cjc=9.728p Mjc=.5776 Vjc=.75

+ Fc=.5 Cje=8.063p Mje=.3677

+ Vje=.75 Tr=33.42n Tf=179.3p

+ Itf=.4 Vtf=4 Xtf=6 Rb=10)

*** Fairchild case=TO92**

Now once again don't be overwhelmed by the magnitude of data involved. Just stick with the standard model libraries or get models from the web. I'll have some pointers at the end of this tutorial, and later I'll have the data on my web page that you can download if you can surf. I'll show all the models and the associated files for this tutorial in the simulations here.

OK, the first thing that we need to do is test the models and look at the characteristic curves for say the PN2222A. What I'll do is show you the file that generates the data and a plot of the data, then I'll explain what is going on.

Here is the SPICE input file that generates the data for the curves for the PN2222A model.


```
GENERATE 2N2222A CE Curve
*
*   2N2222A.cir  Chuck Adams, K7QO
*
*
IB      0 1   DC        1MA
VCE 2   0     DC 12V
Q1      2 1   4 2N2222A
VT      4 0   DC 0
.PRINT DC I(VT)
.DC VCE 0 10V 0.2V IB 0 1MA 200UA
*   model parameters for a 2N2222A
*   transistor
.model 2N2222A NPN(Is=14.34f Xti=3 +         Eg=1.11Vaf=74.03 Bf=255.9
+  Ne=1.307 Ise=14.34f Ikf=.2847
+  Xtb=1.5 Br=6.092 Nc=2 Isc=0
+   Ikr=0 Rc=1 Cjc=7.306p Mjc=.3416
+  Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 +         Vje=.75 Tr=46.91n Tf=411.1p Itf=.6 +
   Vtf=1.7 Xtf=3 Rb=10)
.END
```

Taking the output from the above program and plotting, we get the following graph:

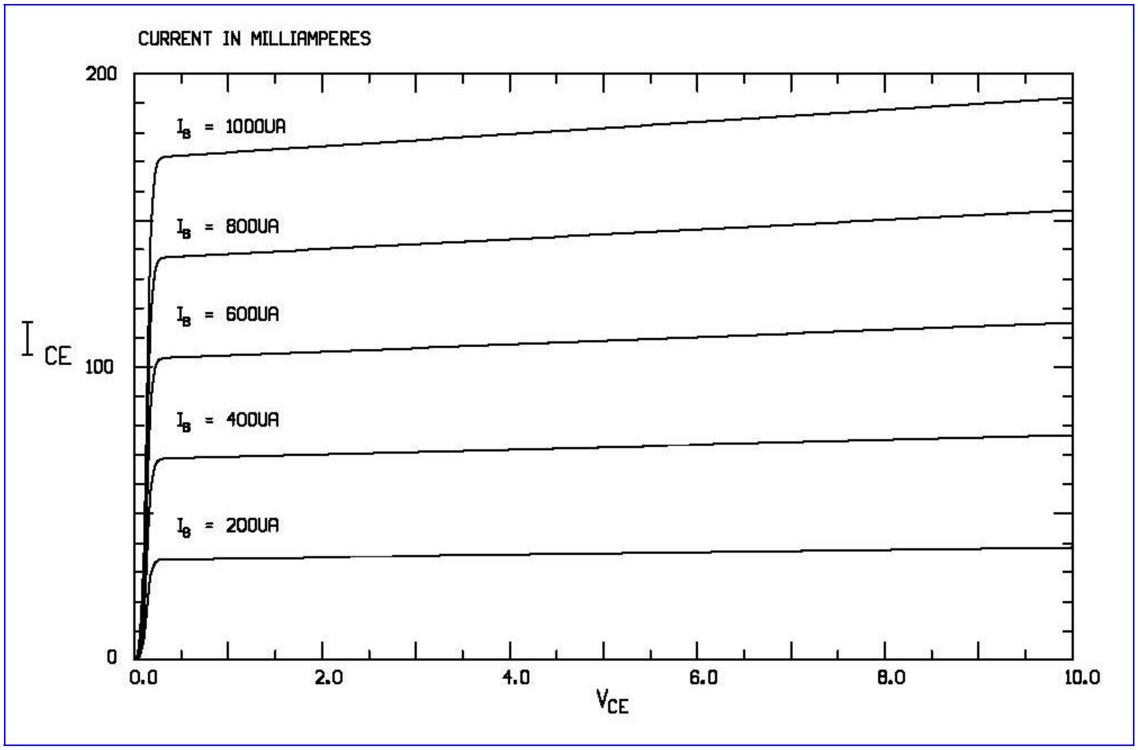


Figure 8. 2N2222A Curves.

What we have done in the SPICE source program is set up a current source IB that will vary from 0 to 1mA in steps of 200µA while we vary V_{CE} from 0 to 10 volts. The good news is that we get the expected curves. The bad news is that SPICE does not tell us that if we do this experiment with a real 2N2222A, it will not survive the high power levels and will almost instantly self-destruct when it reaches a critical power level. So even though SPICE is a great program, it does have its limitations.

MODELING AN IF AMPLIFIER

OK, now we can get down to some actual real modeling of a circuit that we can use in a receiver as an IF amplifier, since we now have everything that we need in SPICE to do it. And since you are relatively new at this game, I would not expect you to develop a circuit from scratch. So, let's go to the ARRL "Solid State Design for the Radio Amateur" written by Wes Hayward, W7ZOI, and Doug Demaw, W1FB, and look at a simple Class A RF amplifier found in Figure 7 on page 21. I am going to modify the circuit slightly to reduce the overall power consumption. Here is the circuit with modification.

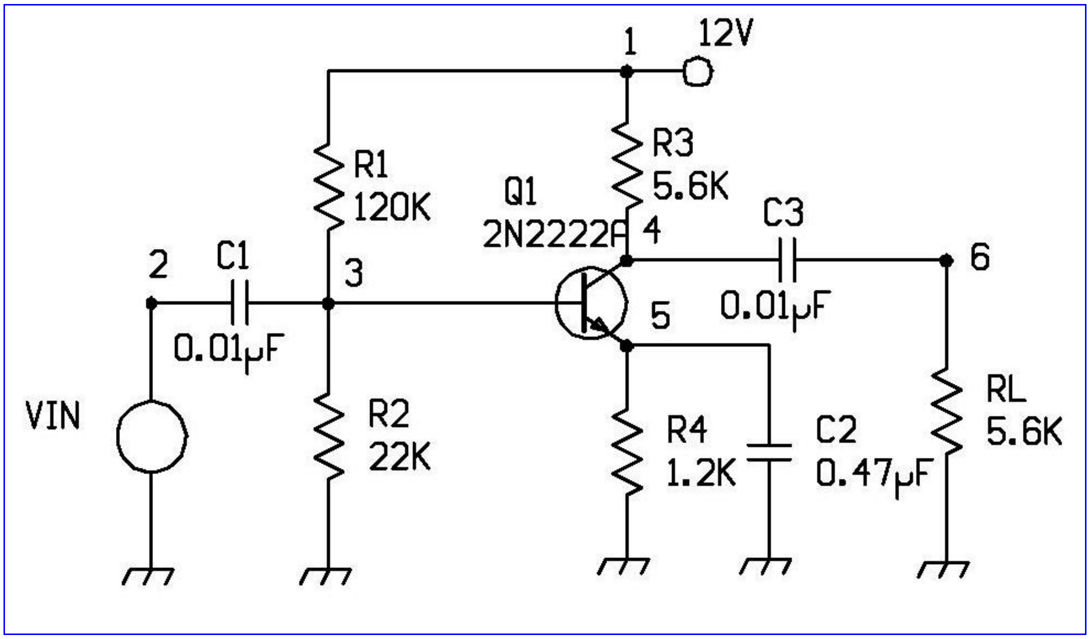


Figure 9. Simple Class A RF amplifier.

Here is the SPICE input file representing the above circuit.

```
Single Stage Amplifier using 2N2222A
*
* Modeled after Wes Hayward's RF
* amplifier
*
* amp2.cir   Chuck Adams, K7QO
*
Vcc  1 0 12
VIN 5  0 AC 1
R1  1 3 120K
R2  3 0 22K
R3  1 2 5.6K
R4  4 0 1.2K
RL  6 0 5.6K
```



```
C1    5 3 0.01uF
C2    4 0 0.47uF
C3    2 6 0.01uF
Q1    2 3 4     Q2N2222A
.AC DEC 10 1E3 1E9
.PRINT AC VDB(6)
.OPTION LIMPTS=5000
.MODEL Q2N2222A NPN(Is=14.34f
+  Xti=3 Eg=1.11 Vaf=74.03 Bf=255.9 +Ne=1.307 Ise=14.34f  Ikf=.2847
+  Xtb=1.5 Br=6.092 Nc=2 Isc=0
+  Ikr=0 Rc=1 Cjc=7.306p Mjc=.3416
+  Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 +        Vje=.75 Tr=46.91n Tf=411.1p Itf=.6 +
   Vtf=1.7 Xtf=3 Rb=10)
.END
```

Running the above simulation in SPICE yields a gain vs frequency plot of:

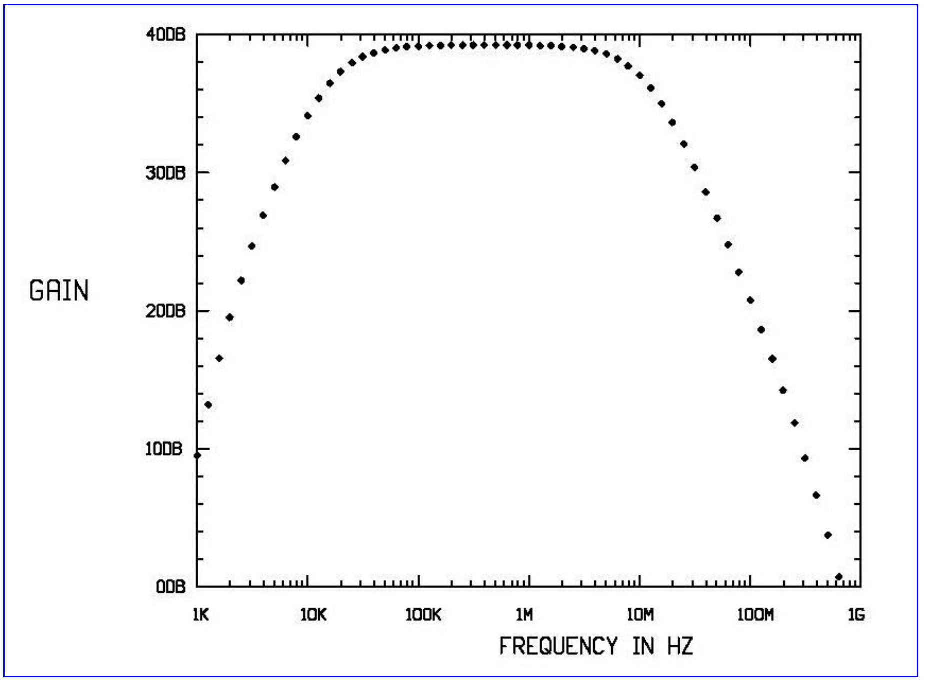


Figure 10. Gain vs Frequency for the single stage amplifier.

In the process of making modifications to the circuit, I managed to reduce the power consumption down from 86mW to 12mW with a change in the resistances and by increasing the value of C2 also increased the bandwidth. C2 will be difficult to get to 0.47µF without an electrolytic but the point is to show that by simple modifications to existing circuits one can experiment and find behavior that would require more time to find on the workbench.

If we were to use the above circuit as an IF amplifier in a receiver we would not require the increased bandwidth and would be happy with the original 0.01µF value. Just to show you how this works, here is the another graph with only the value of C2 changed to a lower value of 0.022µF. At first it does not look encouraging, but remember an IF frequency between 4.0MHz and 10.0MHz the amplifier would do nicely with the gain at least 36dB.

I could spend another three or four pages showing studies that I have made with cascaded amplifiers and a configuration called cascode, but I'd like for you to further your advances by doing these yourself. This purpose of this tutorial is to introduce you to some of the things that can be done and hopefully you will be motivated to study more and use the program yourself.

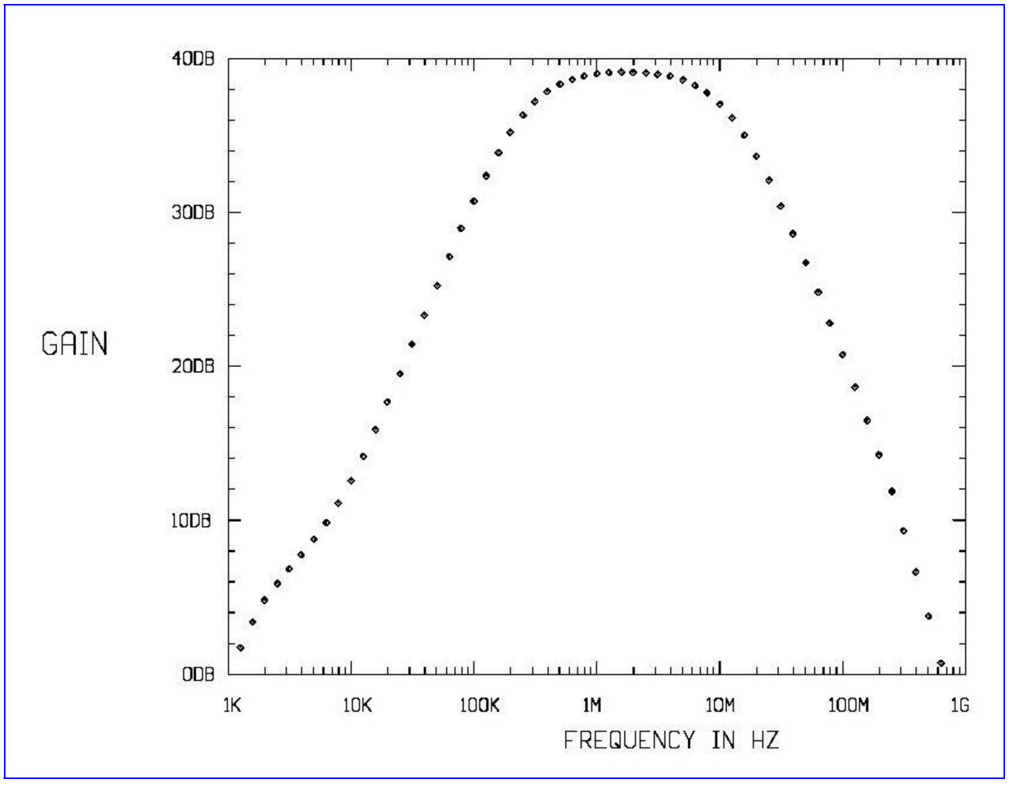


Figure 11. Gain vs Frequency for the modified amplifier.

OSCILLATORS

We have looked at amplifiers, now let's look at an oscillator and we get SPICE to do a simulation. I am going to take just a Colpitts Oscillator and show the schematic, the input file to SPICE, and the output of the oscillator.

First the schematic. This is one that I had shown on my web page for QRP-L on May 1, 1998 during the Elmer101 series. The Elmer101 series was a series of postings by a large number of individuals involved in building the Small Wonder Labs SW-40+ transceiver. In fact, there were over 1,000 postings on the topic in 1998 just to point out how popular the series was. The purpose to get a large group of new builders started with a lot of mentoring from others via the Internet. My personal thanks to Dave Benson, K1SWL, President and founder of Small Wonder Labs, in making it all possible with his effort to rapidly come out with a new kit that helped the project.

So, I took the VFO section of the SW-40+ as shown here and modeled it with some modifications so as not to have to do all the varactor issues. It is presented here as a starting point for experimenters and SPICE modelers to use in modeling other VFOs in the literature or development of a new one. This schematic appeared in the Autumn 1998 issue of QRPp on page 10. Actually I had to go back and redo it as I lost the original file somewhere in juggling between systems at work at the time and when I left. So don't hold me down to the micronic detail (I made that up obviously). This is not rocket science.

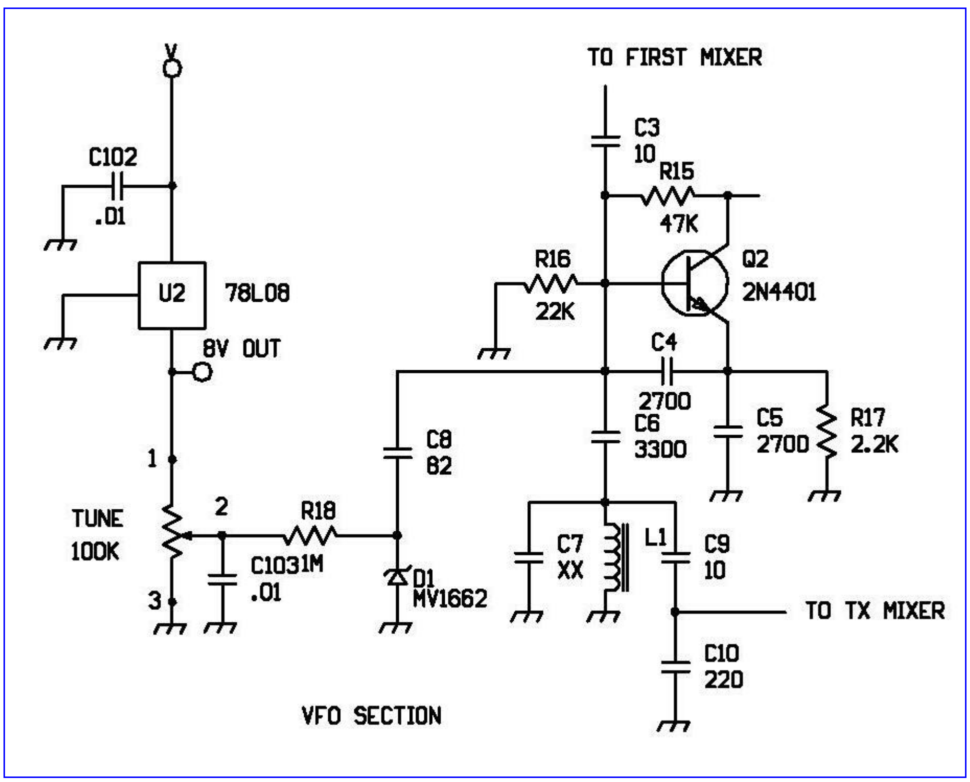


Figure 12. Colpitts Oscillator Schematic.

And from this I created a file for input to SPICE that looks like:

K1SWL Colpitts Oscillator Circuit

```
*   K1SWL Colpitts Oscillator in
*   SW-40+ Transceiver

.model Q2N2222A NPN(Is=14.34f Xti=3 +       Eg=1.11Vaf=74.03 Bf=255.9
+  Ne=1.307 Ise=14.34f  Ikf=.2847
+  Xtb=1.5 Br=6.092 Nc=2 Isc=0
+   Ikr=0 Rc=1 Cjc=7.306p Mjc=.3416
+  Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 +        Vje=.75 Tr=46.91n Tf=411.1p Itf=.6 +
   Vtf=1.7 Xtf=3 Rb=10)

Vcc  2 0 7.5V

C8    1 5 120p
*
*  The following diode represents the
*  varactor
*
CD1  5 0 20p

R60 5  0 100MEG
C6    1 4 3300p
L1 4 0 3.36u
Q1    2 1 3    Q2n2222a
```



```
R15 2  1 47k
R16 1  0 22k
R17 3  0 2.2k
C4    1 3 1300p
C5    3 0 3300p

.TRAN 10n 18u 0 1n
.PRINT TRAN V(4)
.OPTIONS ITL5=50000,LIMPTS=50000
*
.END
```

In the simulation I arbitrarily picked the voltage to display in the following graph and in the .PRINT command in the source file as the top of L1. The purpose here is to show the startup characteristics of the oscillator and the fact that it does resonate quite nicely. Also if you run this simulation on WinSpice3 and expand the display you can determine that the resonant frequency is slightly above 3.0MHz. I like to call it 3.040MHz, since the IF frequency is 4.000MHz and that would put the rig on 7.040MHz. Just for fun.

So here is the graph that I get from the output of the voltage at node 4 at the top of L1 in the schematic.

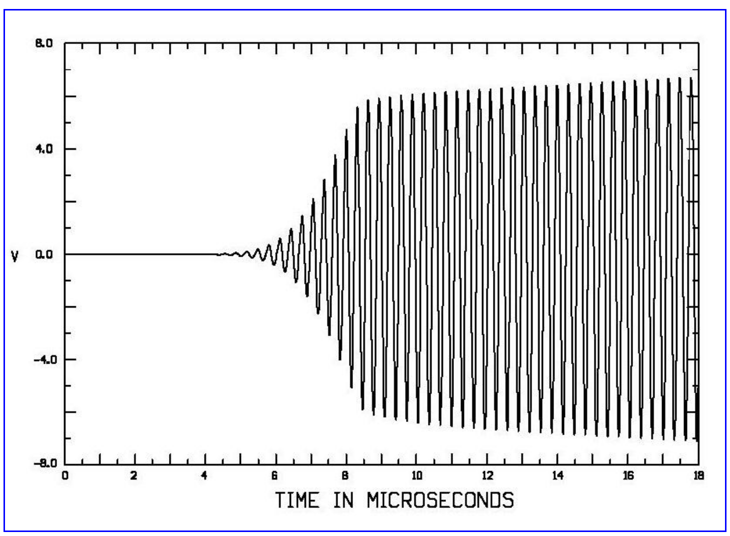


Figure 13. Voltage at L1 in the oscillator.

One of the details that I cannot show here is that if you have SPICE running on a workstation and maybe even on a PC with LINUX all running with the X-window system, SPICE3F4 has an interface whereby you can point and click on anypoint of the graph and get the exact data at that point. But even if you don't have such a system you can use the PRINT command to get all the data and then search for the point that you need. It's all there you just may have to put in a little extra effort to get it.

Because of the nature of this tutorial and a time and space constraint I am limited in just how much material to cover. I have chosen some areas that I hope are of interest and will get you started in the right direction. We are looking at the tip of the iceberg on all the functionality of SPICE. It obviously has a lot to offer but like any large software project you must invest a lot of time and research to become both functional and literate about its capabilities. I read somewhere that it takes a minimum of 40 hours to learn a new software package just to the point of getting comfortable in using it. So it is with SPICE.

I conclude with a list of books that I have in my library that I refuse to part with. Most of the books are out of print but maybe you will run across one at a swap meet or a book store in your travels and will latch onto it. Check with you local community and university libraries as they just might have a copy on the shelves. I don't have a favorite as they each have some new view on a topic not found in any of the other books. Start with the one(s) that you can find on the web bookstores or you can afford. You have my email address. Let me hear from you.

dit dit de K7QO

Books for Reference

This list is not and as I update will never be complete. So many books and so little time.

Banzhaf, Walter., "Computer-Aided Circuit Analysis using SPICE". New Jersey, Prentice Hall. 1989. pp 307 in paperback. Emphasis on SPICE on mainframe computers with ASCII graphics. This of interest to those with old version of SPICE in FORTRAN.

Chattergy, Rahul, "Spicey Circuits - Elements of Computer-Aided Circuit Analysis". Florida, CRC Press. 1992. pp 241 in hardcover. Jury is still out on this book. I need time to work through it. I know the author and the series editor. Small world.

Connelly, J. Alvin, and Pyung Choi, "Macromodeling with SPICE". New Jersey, Prentice Hall. 1992. pp 273 in hardcover. Has floppy with models but 5.25" floppy was unreadable as most likely damaged by magnetic fields with checkout system in the store.

Fenical, L.H., "PSpice - A Tutorial". New Jersey, Prentice Hall. 1992. pp 344 in paperback. A good book for the beginner with plenty of examples with schematics, SPICE output, and graphical output from PSPICE.

Foty, Daniel, "MOSFET Modeling with SPICE". New Jersey, Prentice Hall. 1997. pp 653 in hardcover. With a MSRP of $104, this is by far the most expensive of the lot. Same class as the Massobrio book below in that hams are not the intended audience for this book.

Keown, John, "MicroSim PSpice and Circuit Analysis" .New Jersey , Prentice Hall. 1998 in paperback. With CD evaluation copy of Pspice and Schematic.

Kielkowski, Ron, "Inside SPICE". New York, McGraw-Hill, Inc. 1998 hardcover. With CD with software that is freely distributable and is most likely the same as the one at ftp.lehigh.edu. See notes below on web sources for SPICE and PSpice.

Lamey, Robert, "The Illustrated Guide to PSpice". New York, Delmar Publishers Inc. pp 219 in paperback. With floppy with all examples.

Massobrio, Giuseppe and Paolo Antognetti, "Semiconductor Device Modeling with SPICE". New York, McGraw-Hill, Inc. 1993. pp 479 in hardcover. Written for people in semiconductor manufacturing and their modeling needs.

Monssen, Franz, "MicroSim PSpice with Circuit Analysis". New Jersey , Prentice Hall. pp 548 in paperback. Excellent examples and graphics.

Rashid, Muhammad H., "SPICE for Circuits and Electronics using PSpice". New Jersey , Prentice Hall. 1995. pp 364 in paperback.

Roberts, Gordon W. and Adel S. Sedra, "SPICE Second Edition". New York , Oxford University Press. pp 447 in paperback. I like this book because of the ASCII output from SPICE and graphics and samples with PSpice also.

Tuinenga, Paul W., "SPICE: A Guide to Circuit Simulation And Analysis Using PSpice". New Jersey , Prentice Hall. 1995. pp 288 in paperback. Another good beginner book although weighted heavy to PSpice since author is shown as an employee of MicroSim Corporation which was bought by another company just recently.

Web Sources for Reference

As with all references to the web, be aware that the Internet is a dynamic organism that changes daily. I have seen valuable information disappear almost daily for many reasons. My recommendation is that you check as soon as possible on these sources and download what you think is useful and you have room for on disk drives and other media. Again, this list is also incomplete and using a search engine on the web will give you more references than you have time to browse.

http://www.ticnet.com/k5fo - My home page and I'll try my best to keep links to all this material and more as space and time allows.

http://www.sps-mot.com/home/models/ - Motorola Device Models http://ftp.llp.fu-berline.de/lsoft/Z/1/SPICE.html - links to source code for various systems. Note that some of the links are broken.

http://www.cad.eecs.berkeley.edu/Software/software.html - broken link

http://www.mpce.mq.edu.au/\~tonyp/research/spice.html - good links to source code for version Spice3f4, Macintosh version of SPICE, and links to patches for the software.

http://www.ee.washington.edu/eeca/models/ - links to sources of device and discrete transistor models

http://cbis.ece.drexel.edu/ECE/ECE-E352/vendorsp.html - 20 links to vendor sites with spice models. I didn't follow all the links, but am assuming that they are pretty up to date.

http://www-personal.umd.umich.edu/\~rlab/software.html - links to freeware versions of PSpice for Windows, Macintosh, LINUX, and Commodore Amiga systems.

http://www.ee.washington.edu/eeca/software/ - more links to additional software for filter (analog and digital) and EEDesigner software just to name a few

ftp.lehigh.edu in /pub/listserv/qrp-l/tools - directory are executables for several versions of SPICE including WinSpice3 which I highly recommend.

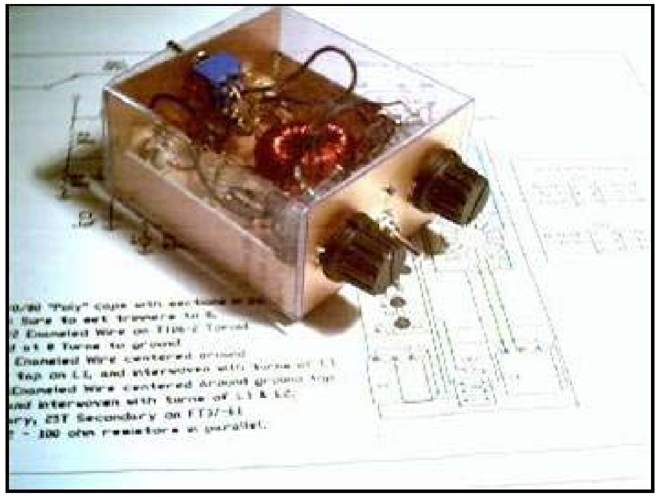
Chuck Adams, K7QO may be reached by mail at Box 11840, Prescott, Arizona 86304-1840, or by email at k7qo@comspeed.net

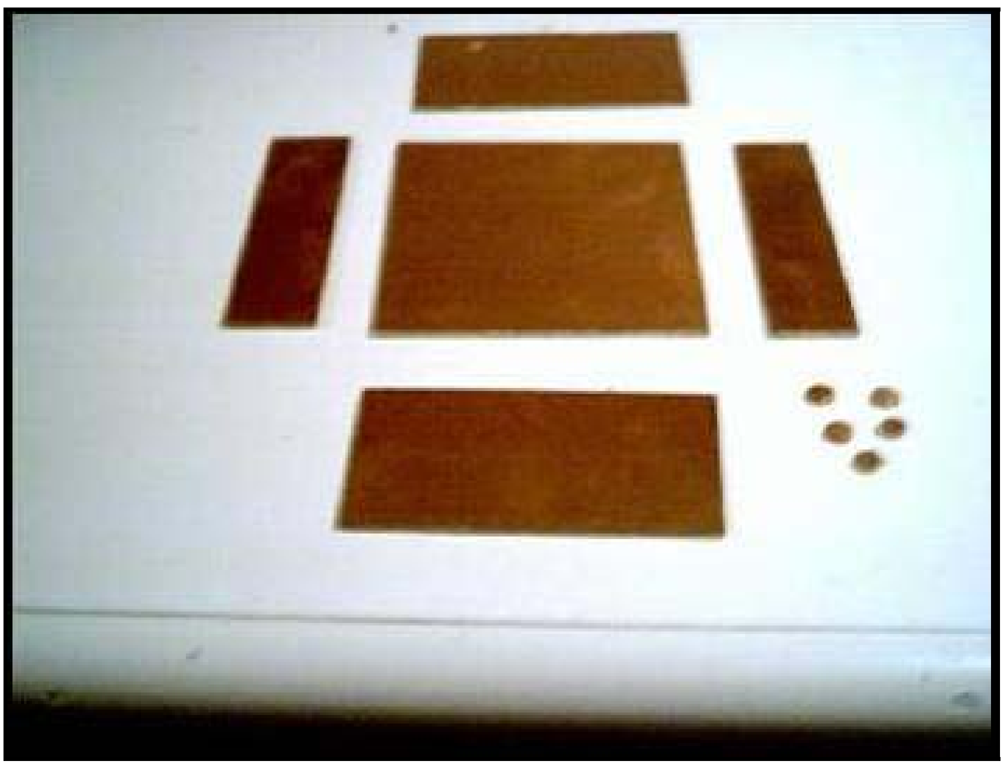
AmQRP.ORG HOMEBREWER

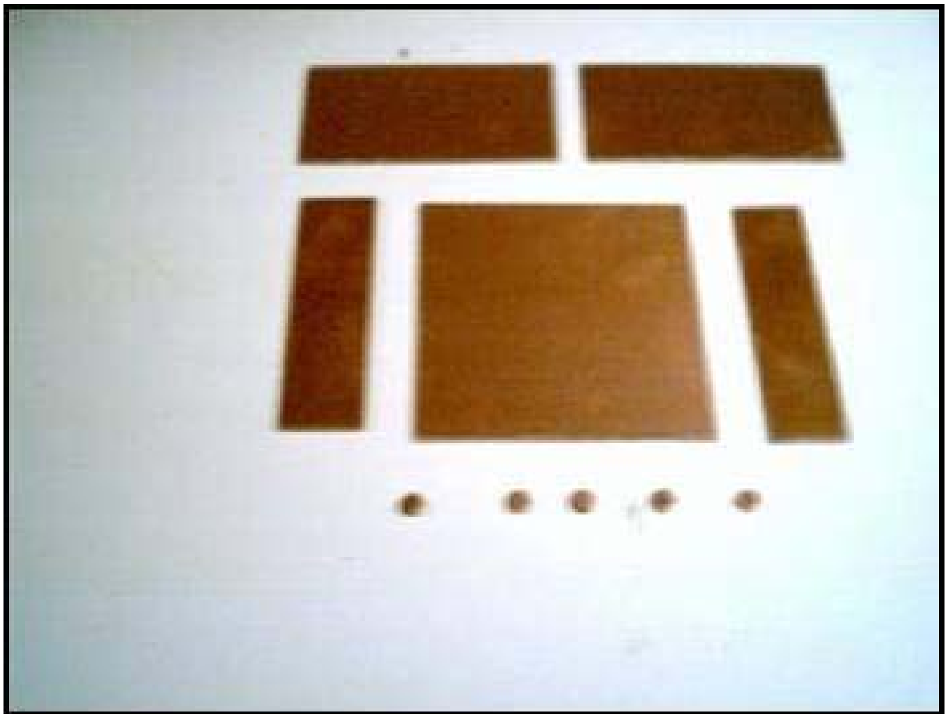
Robert “RC” Conley, KC5WA

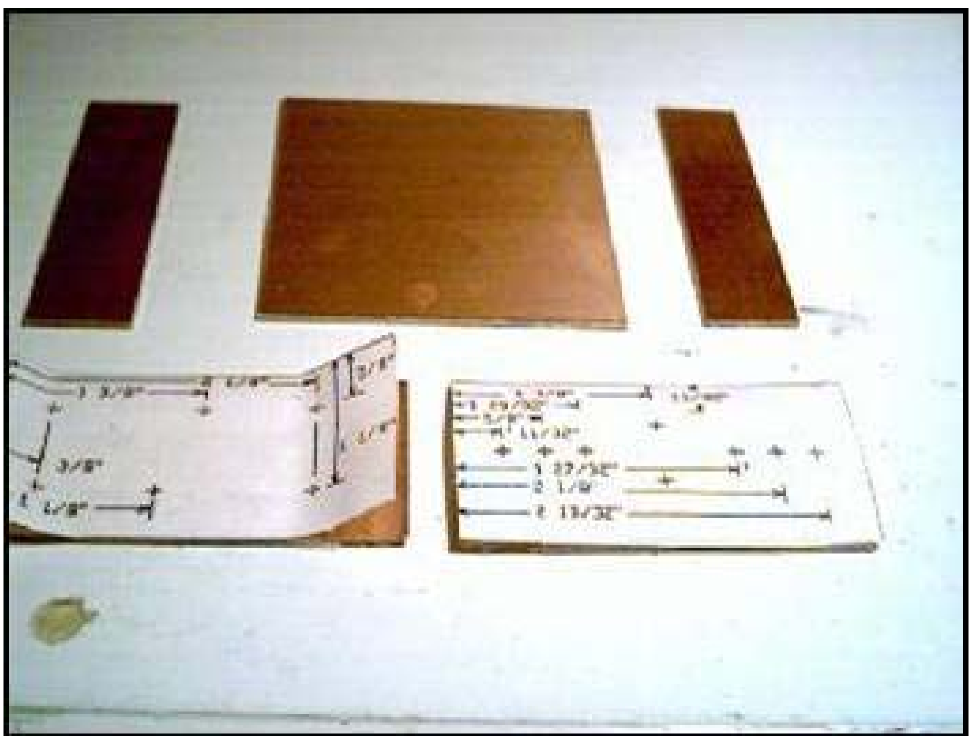
Photo Documenting Construction of the NorCal BLT Tuner

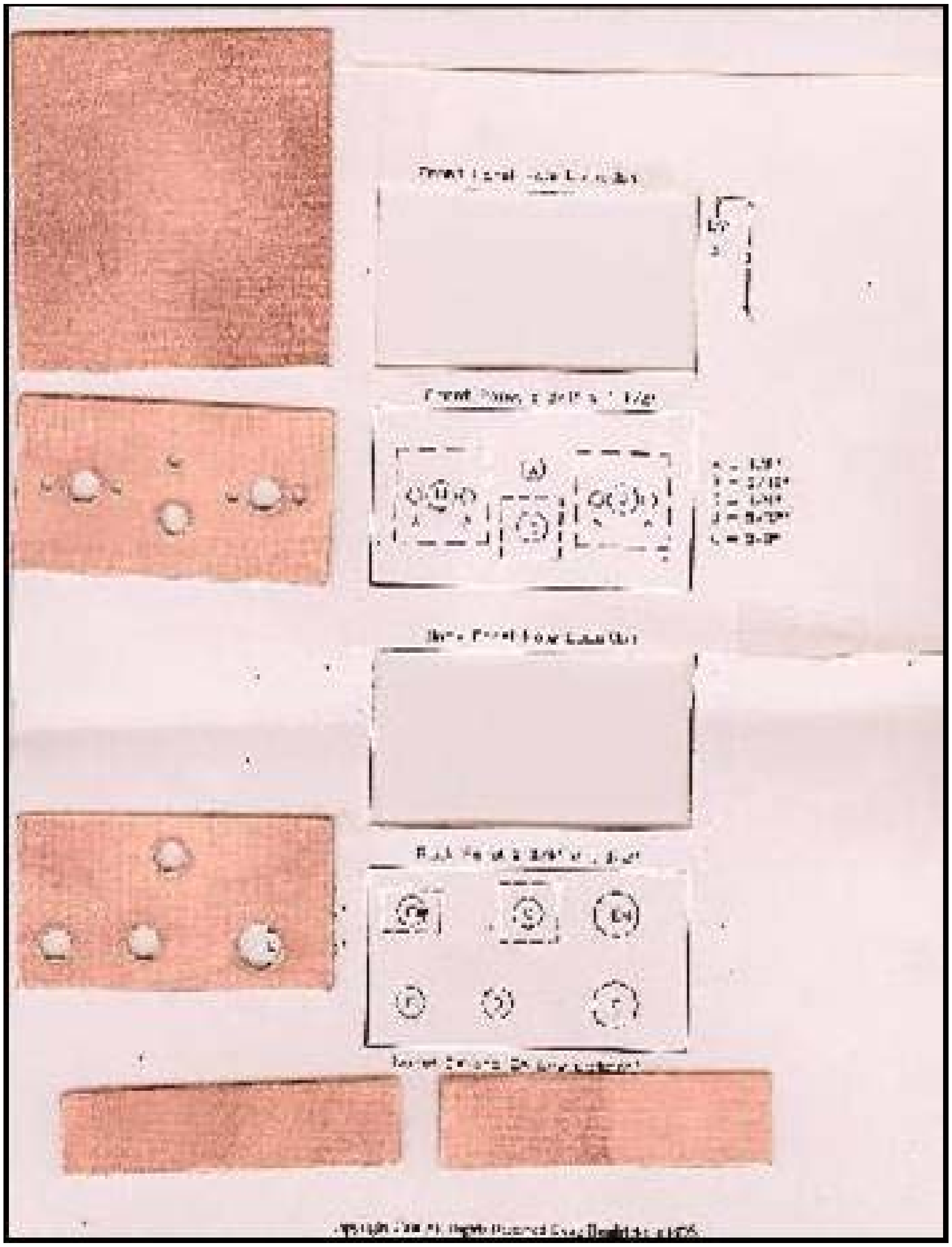
When RC asked if we wanted to see a few photos of how he constructed his original copperclad-enclosed BLT Tuner Kit, we of course said "Sure!". But when he sent over his photo-journalism work, we nearly fell off our work stool ... we think you'll be equally impressed!

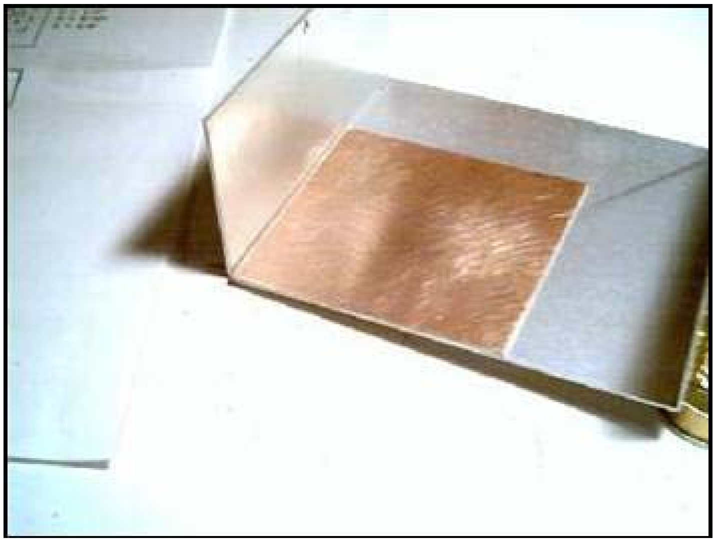


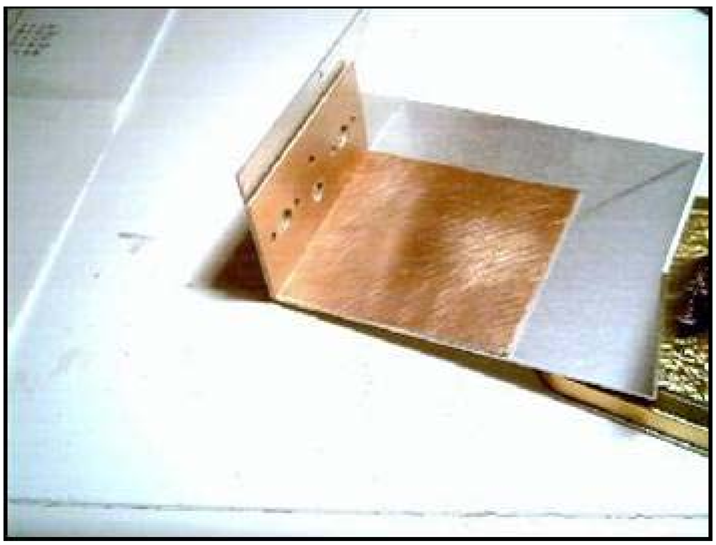


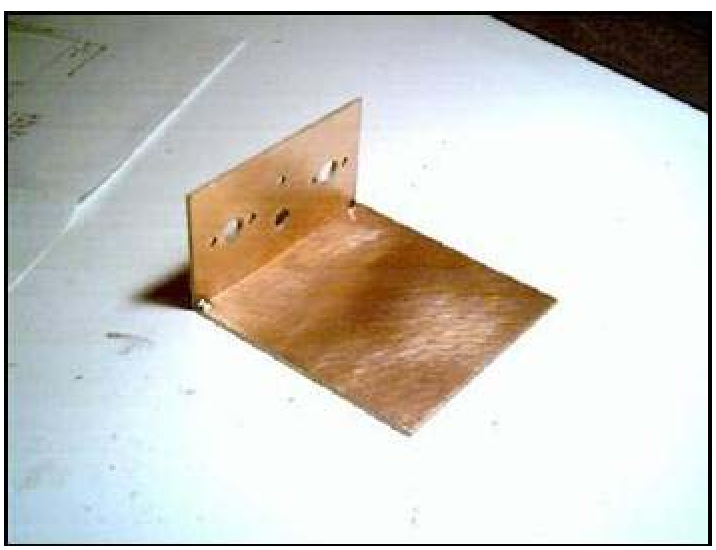


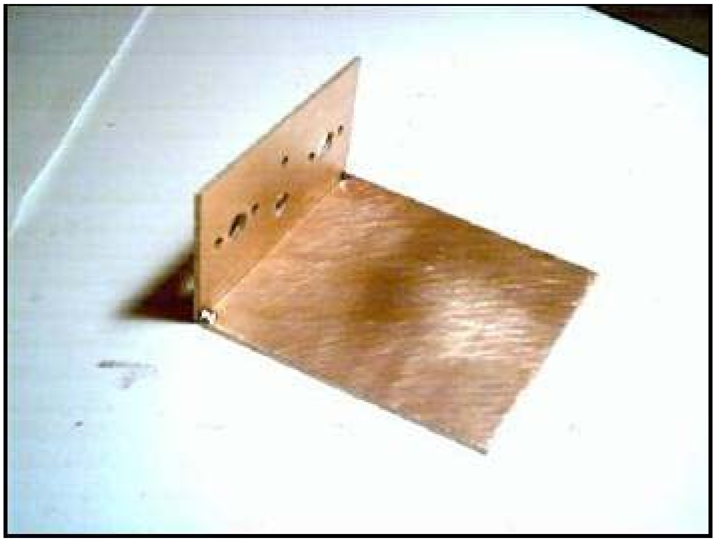

3/8"
1 27/32"
2 13/32"

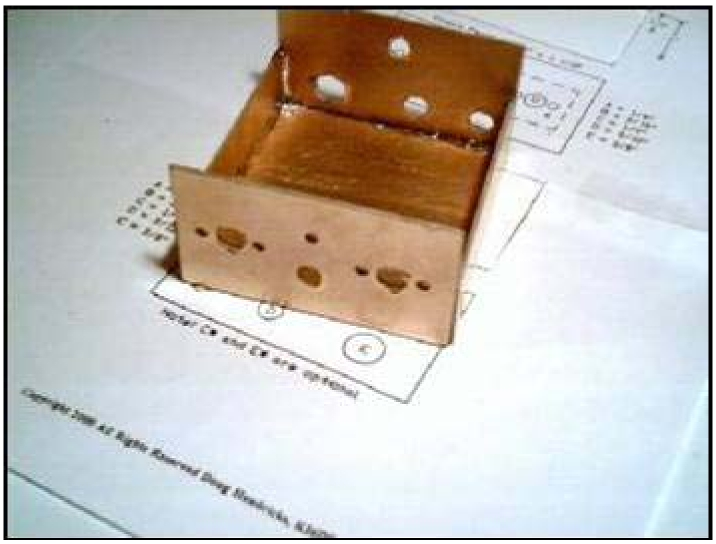


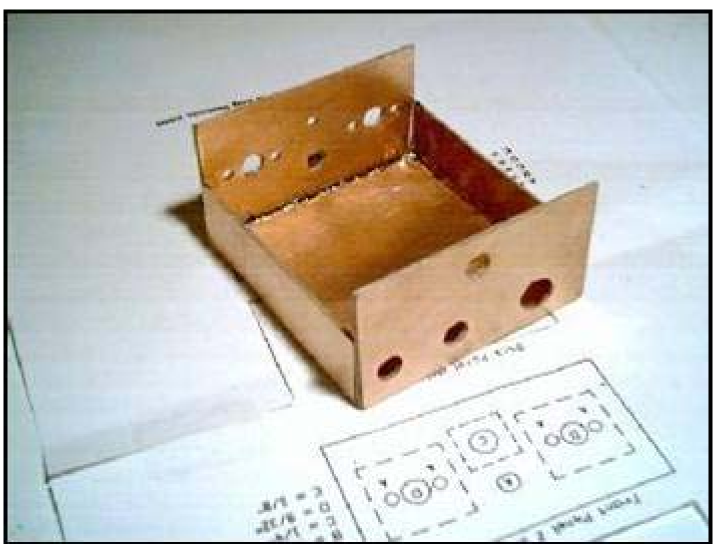


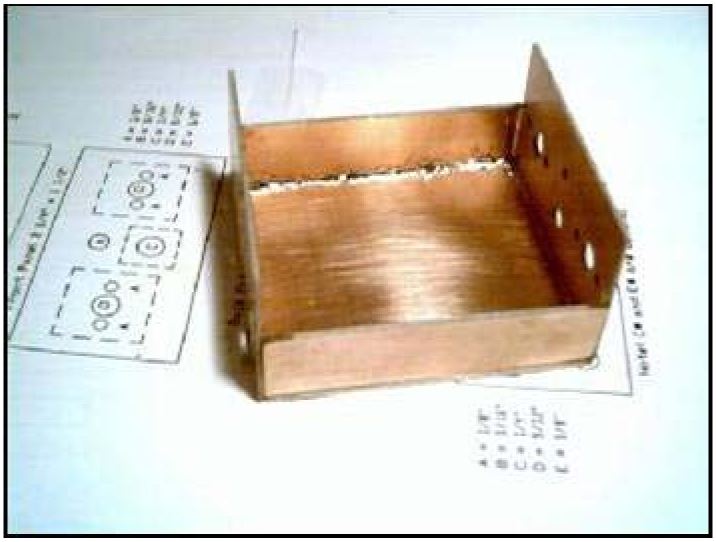


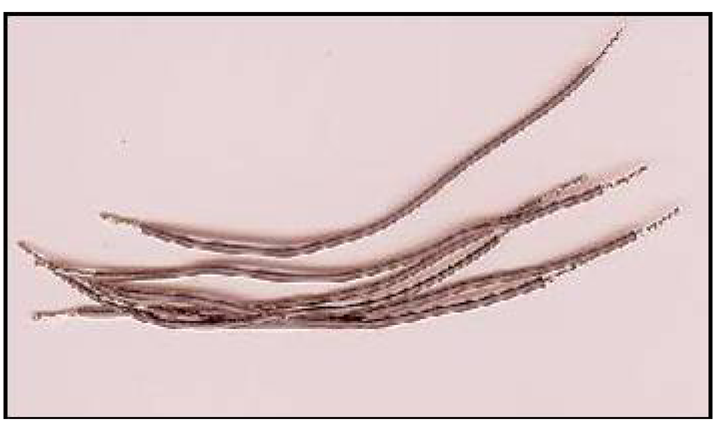


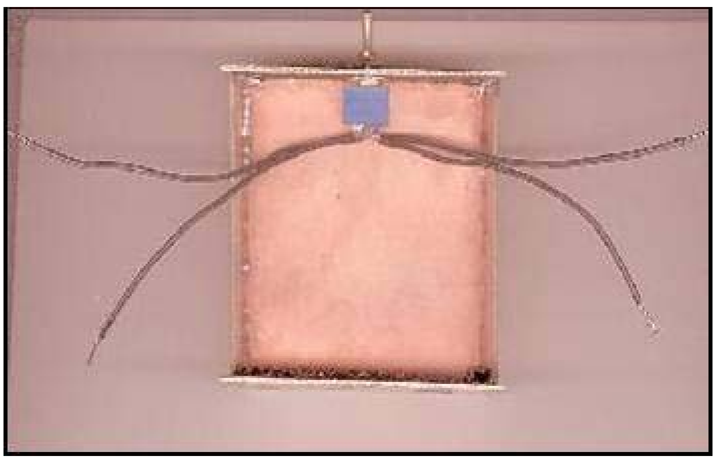


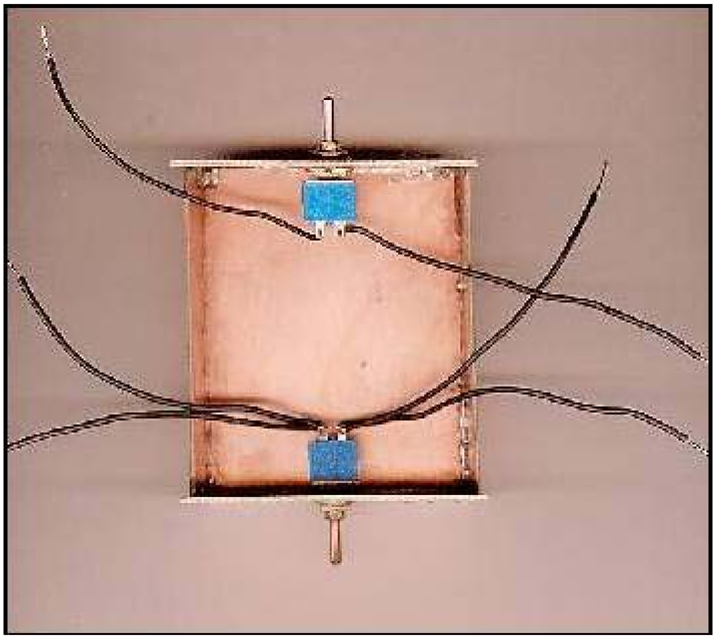


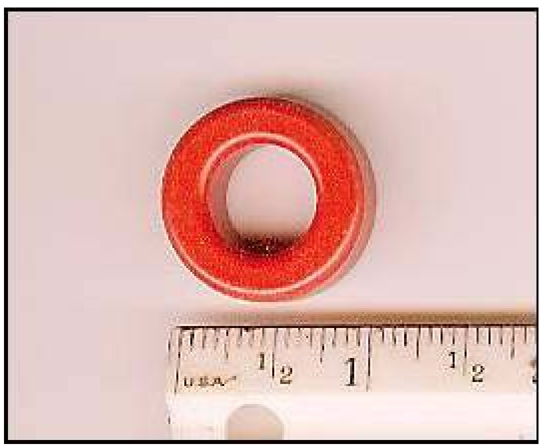


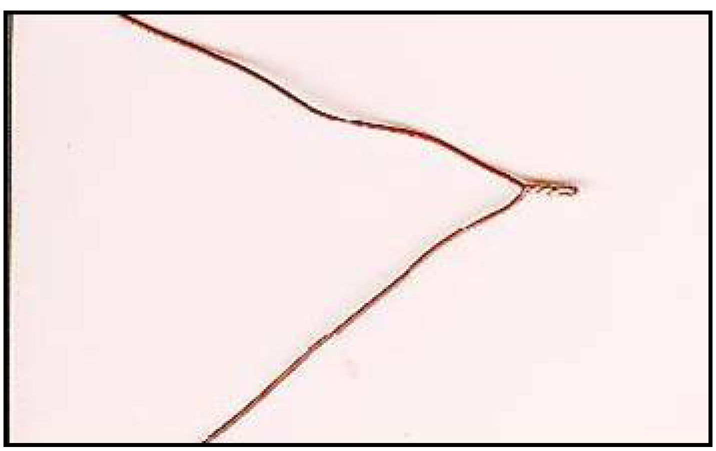


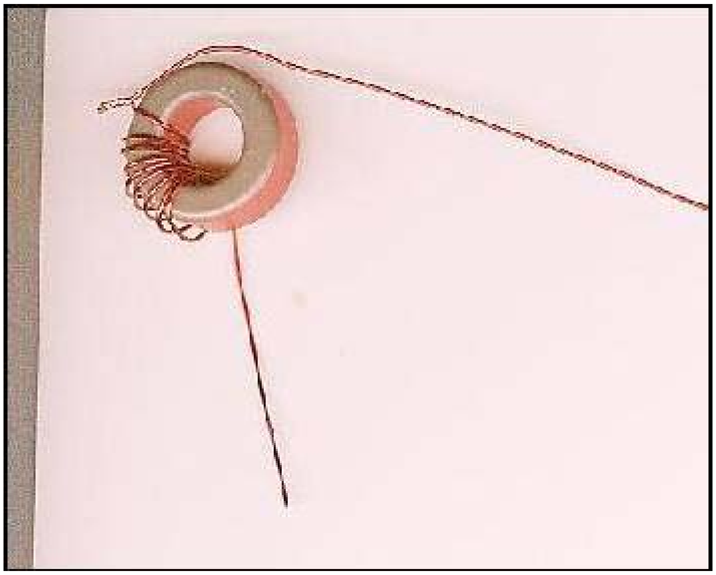


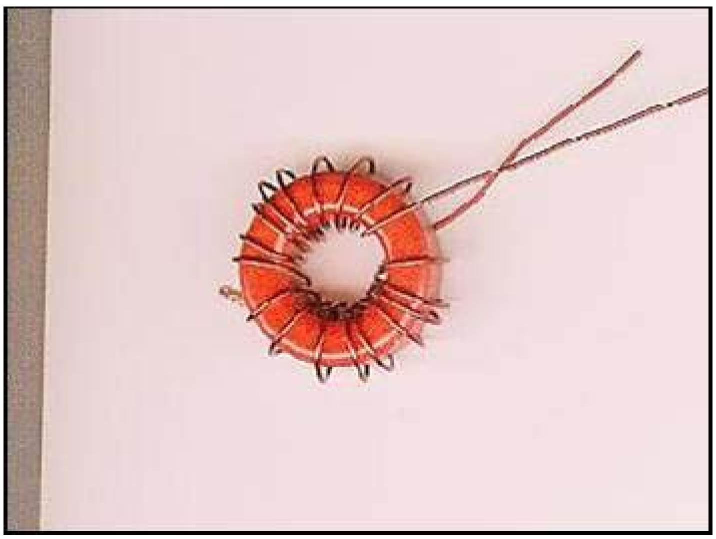


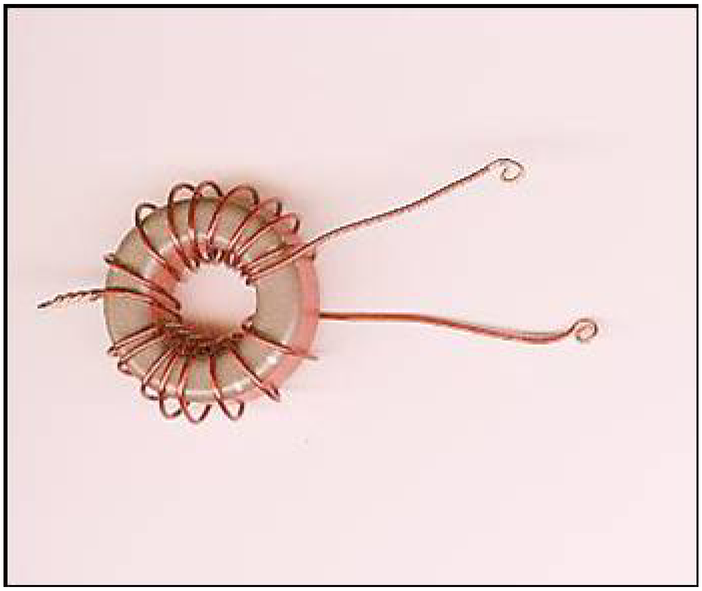


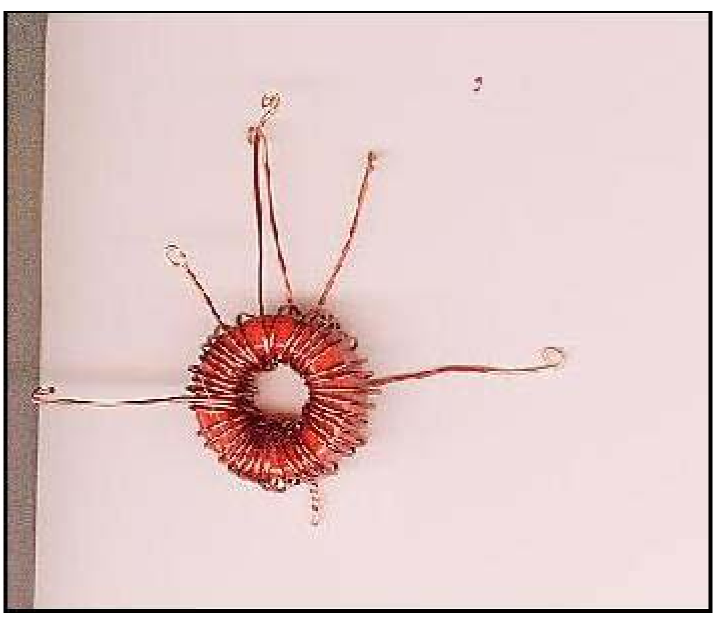


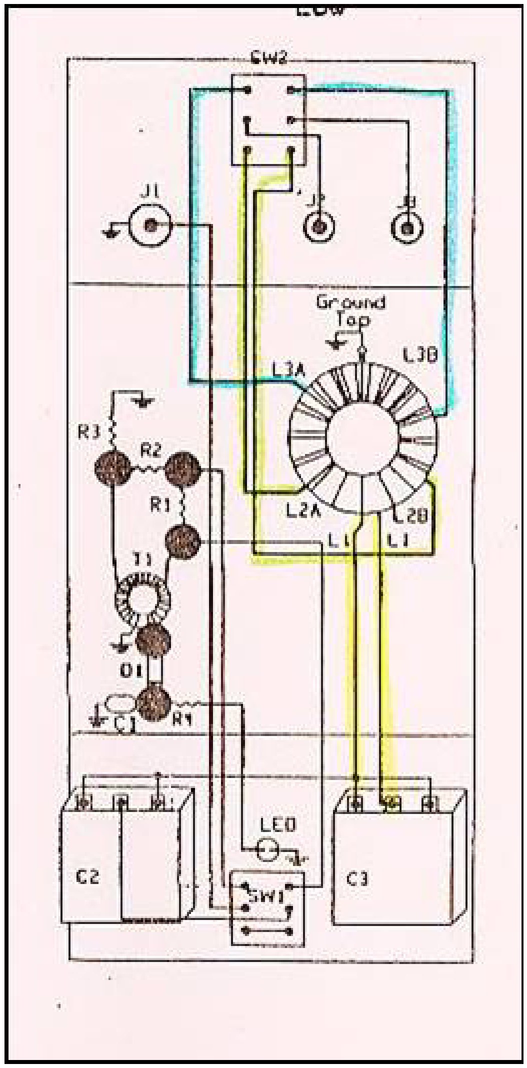


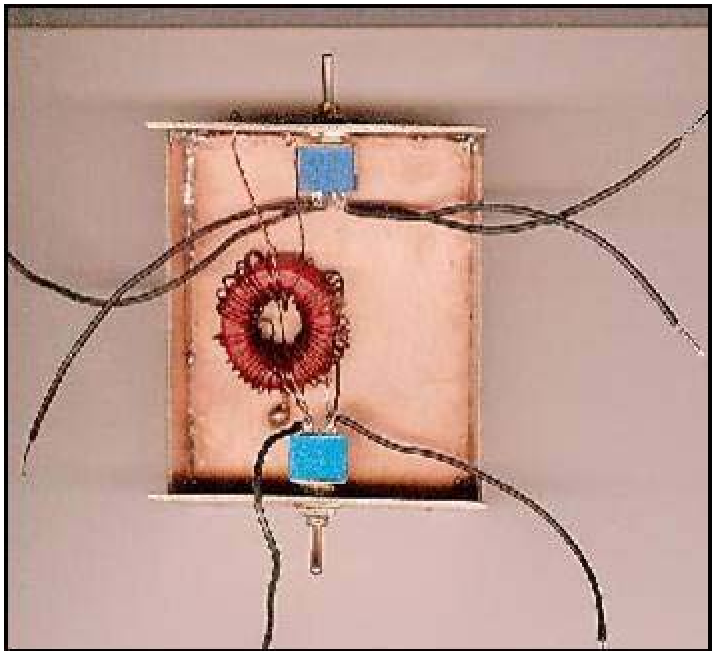


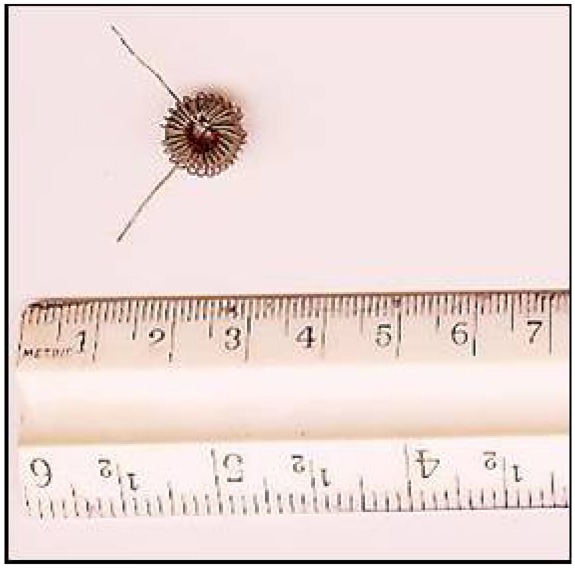


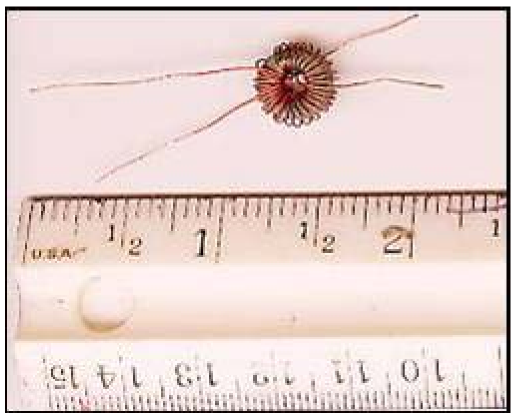


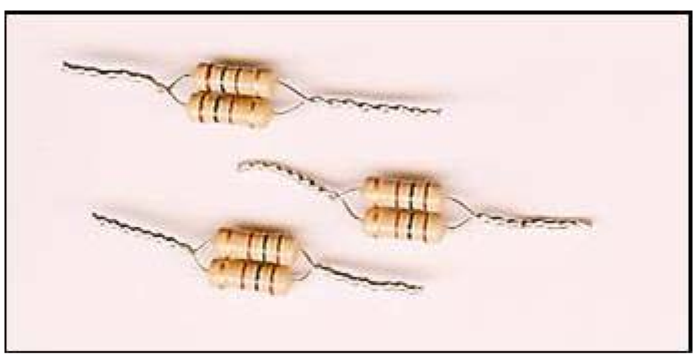


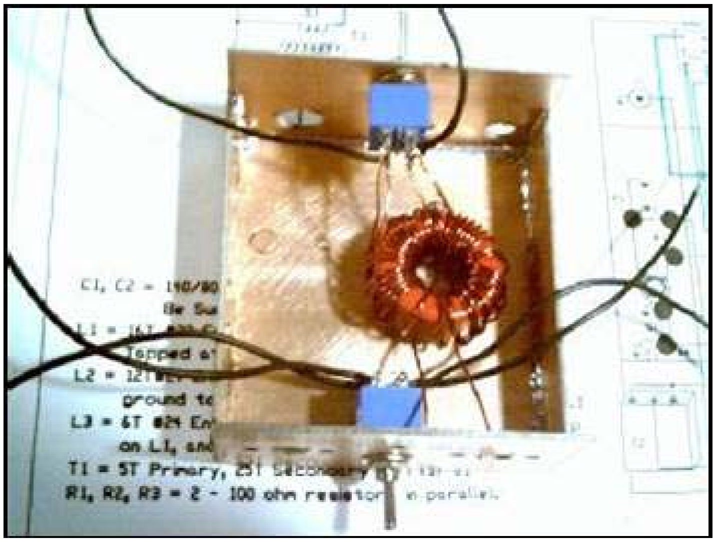

SW2
J1
J2
J3
Ground
Tap
L3A
L3B
R3
R2
R1
L2A
L2B
L1
L1
T1
D1
C1
R4
LED
C2
SW1
C3

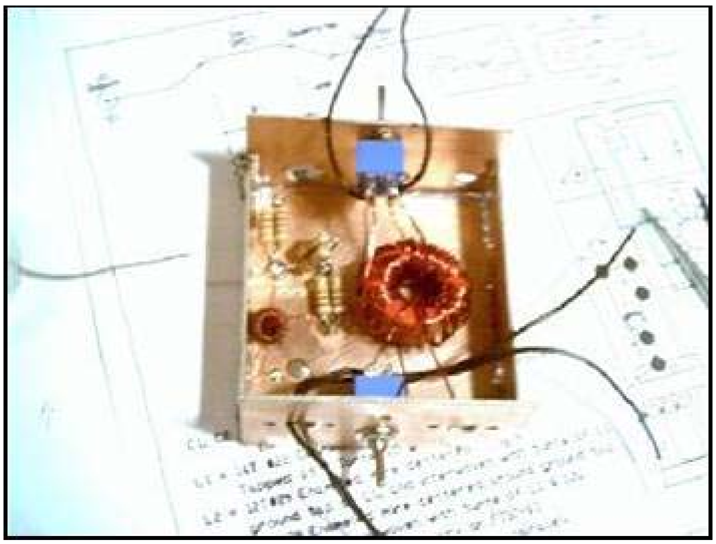


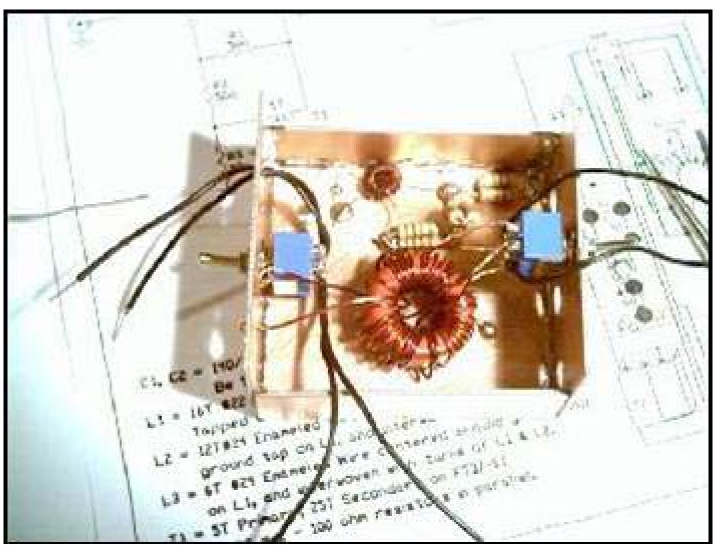


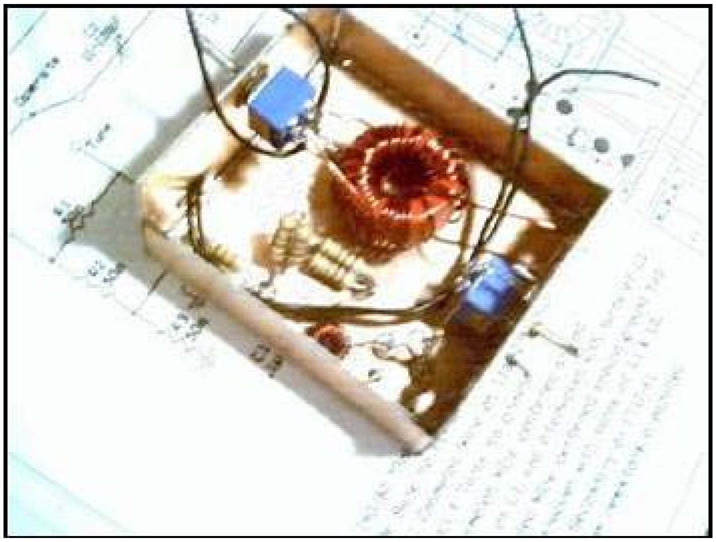


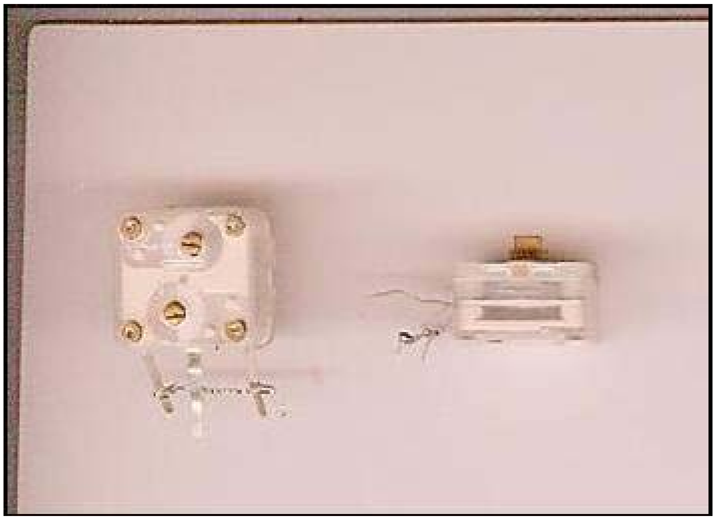


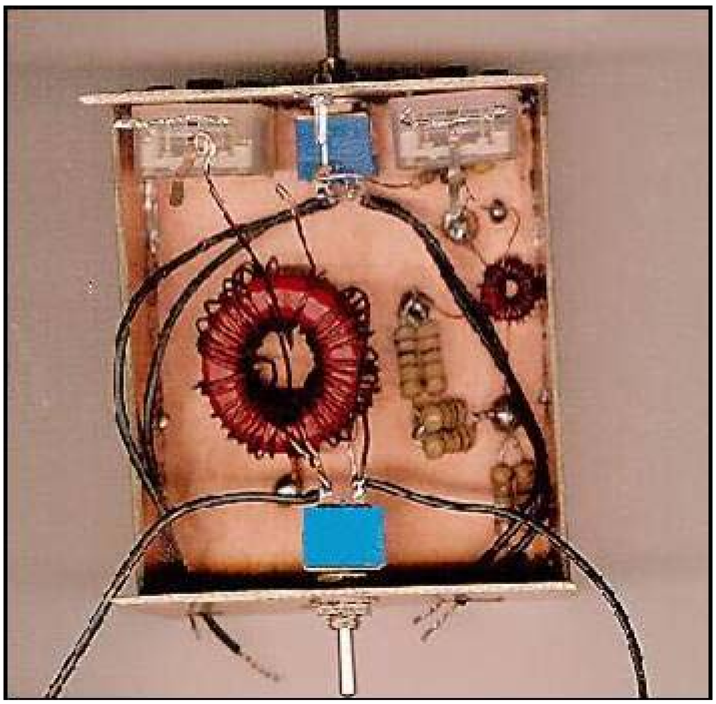

C1, C2 = 140/80
L3 = 6T #24 En
T1 = 5T Primary, 25T Secondary
R1, R2, R3 = 2 - 100 ohm

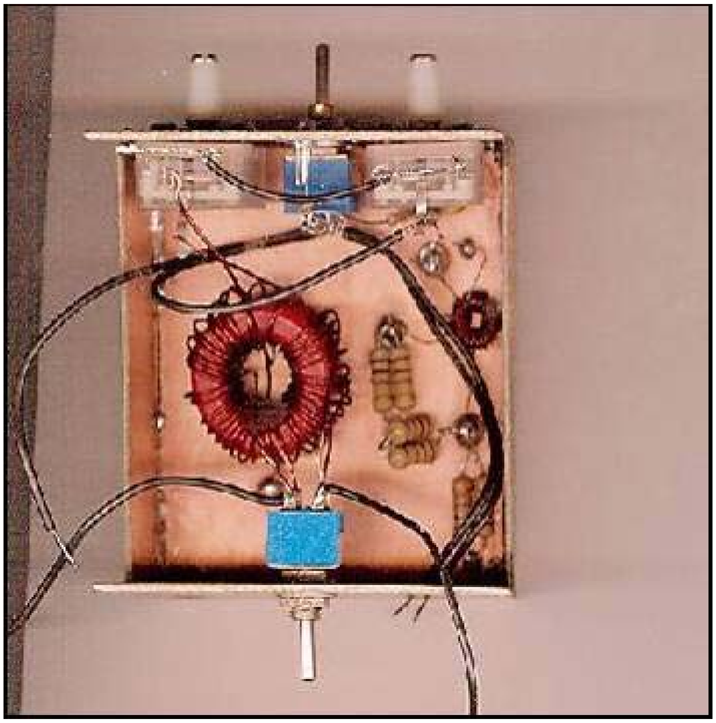


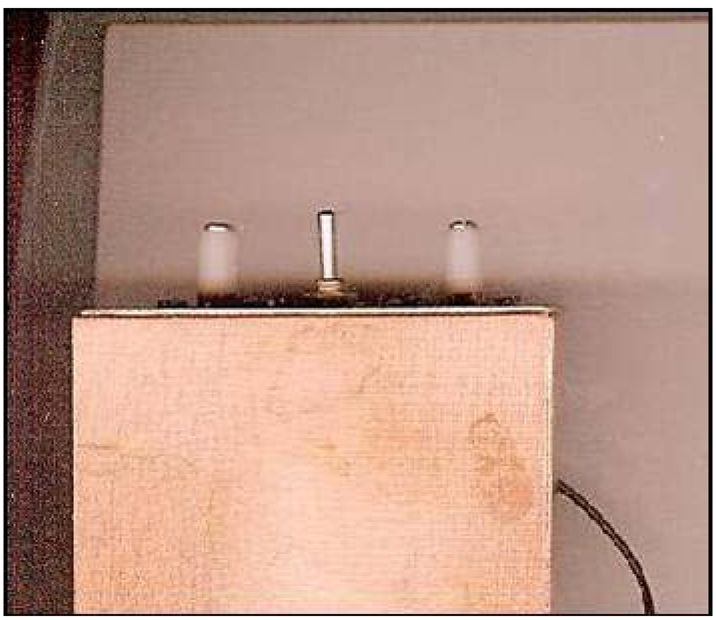


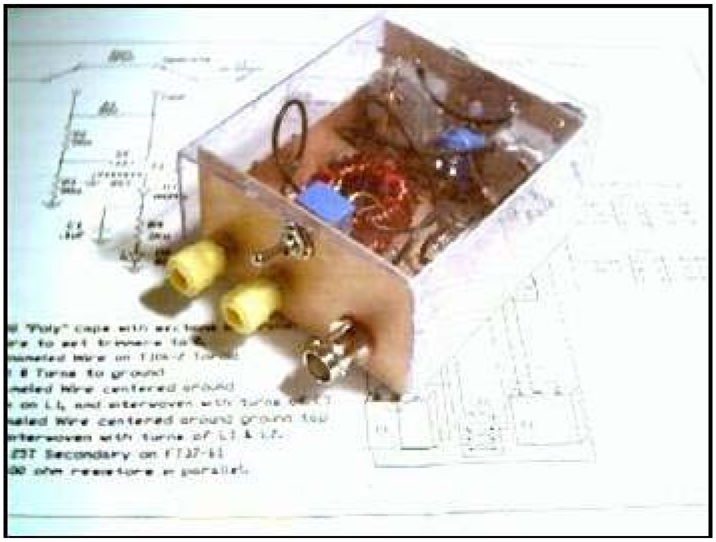


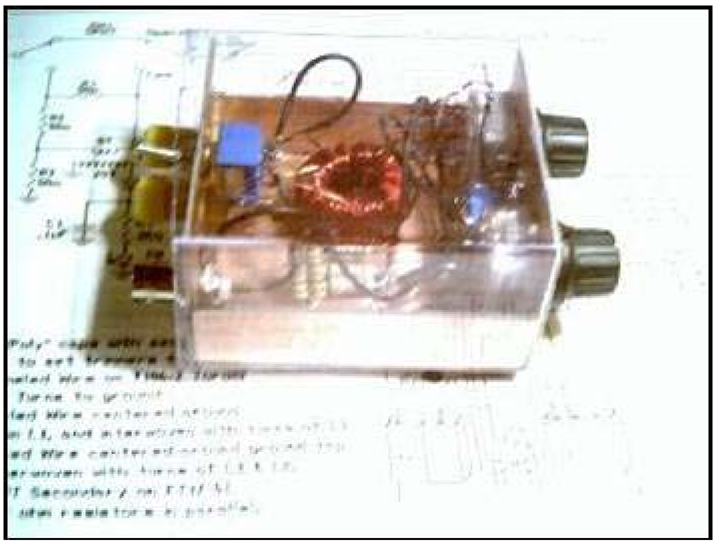


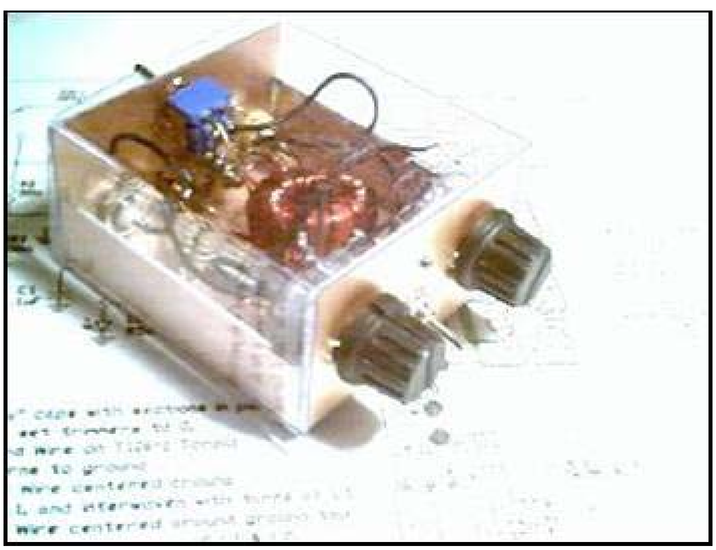


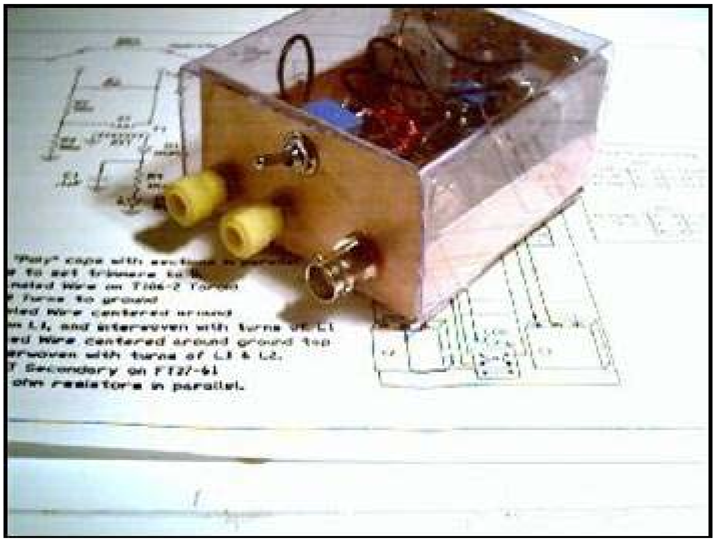


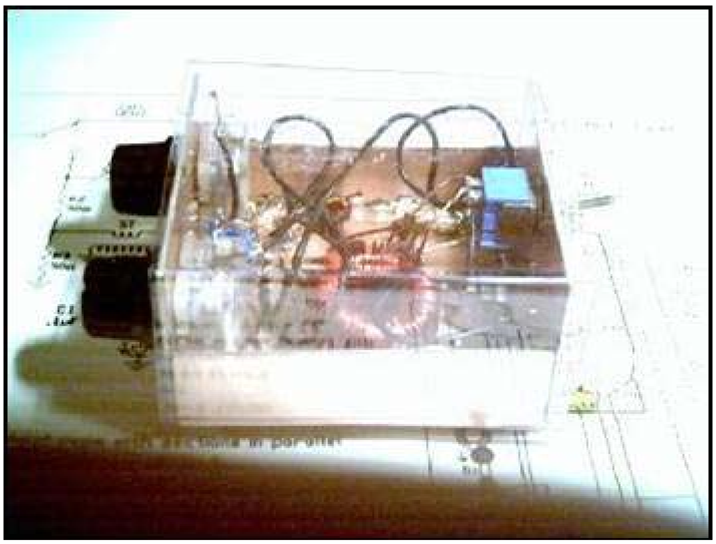


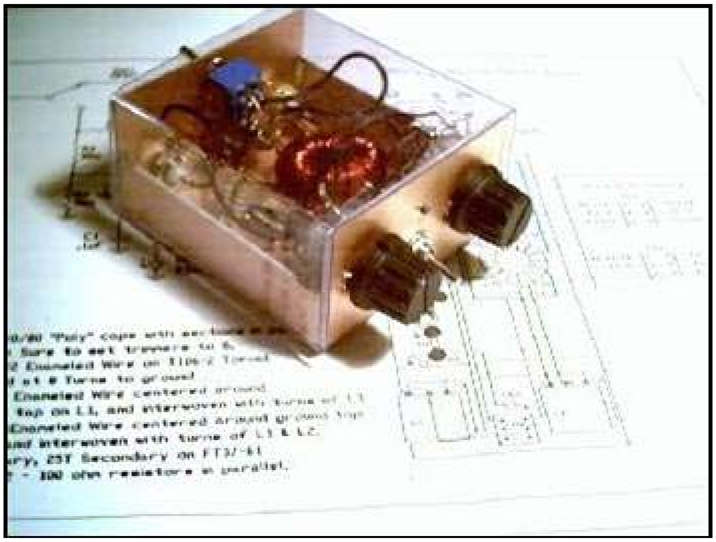












NorCal W6JJZ "BLT" Tuner Kit Manual

Designed by Charlie Lofgren, W6JJZ

Thank you for purchasing a NorCal W6JJZ Balanced Line Tuner Kit. The tuner was designed by Charlie Lofgren, W6JJZ, who is renowned in the QRP World as a tuner expert. Charlie has built all of the tuners used by the Zuni Loop QRP Expeditionary Force for years, and they all swear by them.

This tuner is a balanced line tuner only, and will not work with coax feedlines unless modified as shown in the mods and improvement section.. But it works great with open wire feeder, ladder line, zip cord, and ribbon cable. As long as you are using balanced line as a feedline, this tuner will work.

Charlie designed this tuner to work specifically with the polyvaricon variable capacitors available from Mouser. I asked him to design it at first because I wanted a simple tuner for a presentation that I was doing at the Ft. Smith QRP Group Forum, ArkieCon 2000. It turned out so well that everyone who saw it wanted one. Thus the NorCal W6JJZ BLT Kit was born. I would like to thank Charlie for his efforts on behalf of NorCal. This one is going to be a classic.

The design is for a classic Z match, using inductive coupling with L1, L2 and L3 wound on a single T106-2 toroid. L2 or L3 is switched in and out of the circuit by Switch 2, located on the back panel of the tuner. The "high" and "low" positions on the switch for the output links may need clarification. The positions are for "high" and "low" in terms of impedance, not frequency. For a given band and antenna, try the high Z link first, and use the low Z link only if a match can't be found with the high link. (Often either link will allow a match. In these instances, the high Z link produces better efficiency as a result of loading the tank circuit more heavily.)

The circuit also includes the famous N7VE LED SWR indicator circuit. Dan Tayloe invented this several years ago, and it has proven a great addition to the qrp fraternity. This allows us to have an indication of lowest SWR on the tuner (indicated by dimming or LED going out at minimum SWR).

The circuit also is an absorptive bridge, which means that your transmitter sees a 50 ohm load as you are tuning up, which will help to save your finals. This tuner is rated at 5 Watts. I doubt if the polyvaricon caps will take the 100 Watts of your big rig!!

The first step in building the rig is to build the custom case. The case is made out of .060 pcboard stock, and has been precut to size for you. All that you have to do is drill the holes for the front and rear panels, and then solder the kit together.

If you have tried to build a case out of pc board and had trouble keeping the sides square, you are not alone. But George Heron, of the NJ QRP Club has figured out the secret of building these cases and shares his secrets here.

The parts for the case have been mass produced. You need to check them to make sure that you will have a good fit. Start with the sides and the bottom. They should be exactly the same length. If not, file to size. Also, be sure that you keep it square. Next, check the front and back panels, make sure that they are the same width as the bottom, if not mark and file as before.

Next, use the drilling templates to mark the holes as indicated and drill the front and back panels to the size indicated. Make sure that you are accurate on the capacitor holes, as you do not want to have the center shaft of the capacitor touching the case.

Now that you have the holes drilled we are ready to start the case assembly. Take the bottom piece and one of the side rails. Line up the side rail with the ends flush and the side on the bottom so that the edge of the bottom is flush with the outside surface of he side. Tack solder it on one end. Get it as straight as you can, but don't worry about it, we will adjust it later. What we are concerned about is that the ends are flush. Now, place the front panel against the ends of the bottom and the side panel. Make sure that it is oriented correctly, lined up, and solder it on one end. Check your work. If everything is lined up, unsolder the side rail and make it fit straight, flush with the edge of the front panel. Now, solder the bottom of the side rail and the junction of the top side of the side rail and the front panel. Do the other side rail as you did the first. This will hold the front panel square, and now you may tilt it up to solder the seam.

When you finish the front panel, do the same with the back, making sure that you orient the panel correctly. The case is really quite easy to build and the neat thing about it is that you use the parts to self align it. Many thanks to George Heron, N2APB for his invaluable assistance with these instructions. In other words, George, thanks for the trade secret, grin.

The last step with the case is to fit the Lexan plastic top to the case you have built. Place the top over the case, and secure with tap. Drill a 1/8" hole in the middle of the side rail. Repeat on the other side. Make sure that your top is square before you drill the holes. Drill them through the plastic and the case.

Use a 4/40 x 1/4" Machine screw to attach the top. You will need to place the screw through the hole of the case, then tighten a brass 4-40 nut on the inside of the side. Solder the nut to the inside. Instant pem nut, grin. Remove the screw, let cool, and place the top on to make sure it fits. The top can be trimmed with ordinary scissors if you wish to have a tapered front.

Now we are ready to start building the tuner. The first step is to prepare 6 wires that are 4" long. Solder 4 wires to Switch 1 as shown in the wiring diagram along with the jumper. Mount Switch 1 in the front panel. Next, solder two wires to the middle connections of Switch 2. Mount Switch 2 in the back panel.

We are ready to wind the two toroids now. Start with the larger one, the T106-2. Cut off 30" of the heavy red enameled wire. Bend it in half, and clean off the insulation for 1/2" on either side of the center of the wire. Then, twist the wires together 3 times forming a tiny loop that will be the center tap of L1. Take the toroid, put the wire through the center of the toroid and hold the twisted loop against the edge. Wind the wire 8 times around the toroid, counting 1 turn each time the wire goes through the center of the toroid. Now, wind the other end of the wire 8 more times going in the opposite direction on the toroid, but the same direction in winding sense. When you finish, spread the turns evenly around the toroid, and bring the ends of the wire up to the side opposite the twisted loop, clean the insulation from the ends and make a small loop to solder to.

The coils for L2 and L3 are wound interspersed and in the same direction as L1. Use the heavy wire provided. The same wire that you used for L1. The wire for L2 is 24" long, and the wire for L3 is 12". Place the toroid in the case approximately where it will be and trim the wires to the lengths needed to attach L2 and L3 to Switch 1 and to attach L1 to C3 as shown in the wiring diagram. Clean off the insulation by burning it back for about 1/2" and then carefully scraping off the residue with a knife. Now you are ready to mount the big toroid. Place it where it goes in the case, and solder the tap of L1 to the bottom of the case. Then solder L2 and L3 wires to Switch 2. We will solder the connections to C3 later.

Now lets wind the smaller toroid. You have two smaller diameter pieces of wire, one red and one green. Start with the red wire and wind 25 turns on the toroid. Count the turns on the inside of the toroid with each time the wire passes through the center of the toroid as one turn. Trim the wire to 1 inch leads, remove the insulation and tin the leads. Now, take the piece of green wire and wind 5 turns. Start the winding in the middle of the red wire. Trim the ends of the green wire to 1", remove the insulation and tin the leads. Set the toroid aside for now.

Prepare the six 100 ohm 2 watt resistors by twisting the leads together to make 3 pairs of resistors. This will result in 3 - 50 ohm resistors.

Now we are ready to build the SWR absorptive bridge and LED indicator circuit.

Use the layout drawing as a guide and build the circuit Manhattan style. You will find several round pads provided in the kit. Use these as "tie points" as shown. They are glued down to the base of the case using super glue. One small drop per pad is plenty. Place the drop where you want the pad, then with tweezers or needle nose pliers place the pad on the glue. Press down and hold for 30 seconds. Tin the pad. When you have the pads in place, build the circuit. The ground symbol means that you solder the end of the component to the base of the case, which is ground. Make sure that you orient the diodes correctly. The LED has the short lead grounded to the case front which holds it in place, or you may use a drop of super glue here, just be careful to not get it on the lens. The other lead of the LED connects to R4, the 1K resistor.

Now we will prepare C2 and C3. Put a jumper between the outside leads of each capacitor as shown. Then, make sure that the trimmer adjustment caps on the back are set at lowest capacitance, which means fully unmeshed. Now mount the two caps to the front panel. Make sure that the center conductor does not touch the case, or your tuner will not work right. Tighten the small screws snug, but be careful to not strip the threads. Run a jumper wire from the two tied together connectors on C2 to the two tied together connectors on C3. Connect the middle connector of C2 to SW1 as shown.

The caps do not have shafts for knobs, but we can fix that easily. In your kit of parts you will find two nylon spacers. Attach them to the center hole of the cap with the 2.5 x 16mm screws provided. You will want to put a drop of super glue between the spacer and the cap to keep it from spinning, or your could use a tiny lock washer here (not provided). Just be careful to not get any glue in the cap!! Now you have a shaft to attach the knobs to!! Thanks to Dave Gauding, NF0R who showed me that trick.

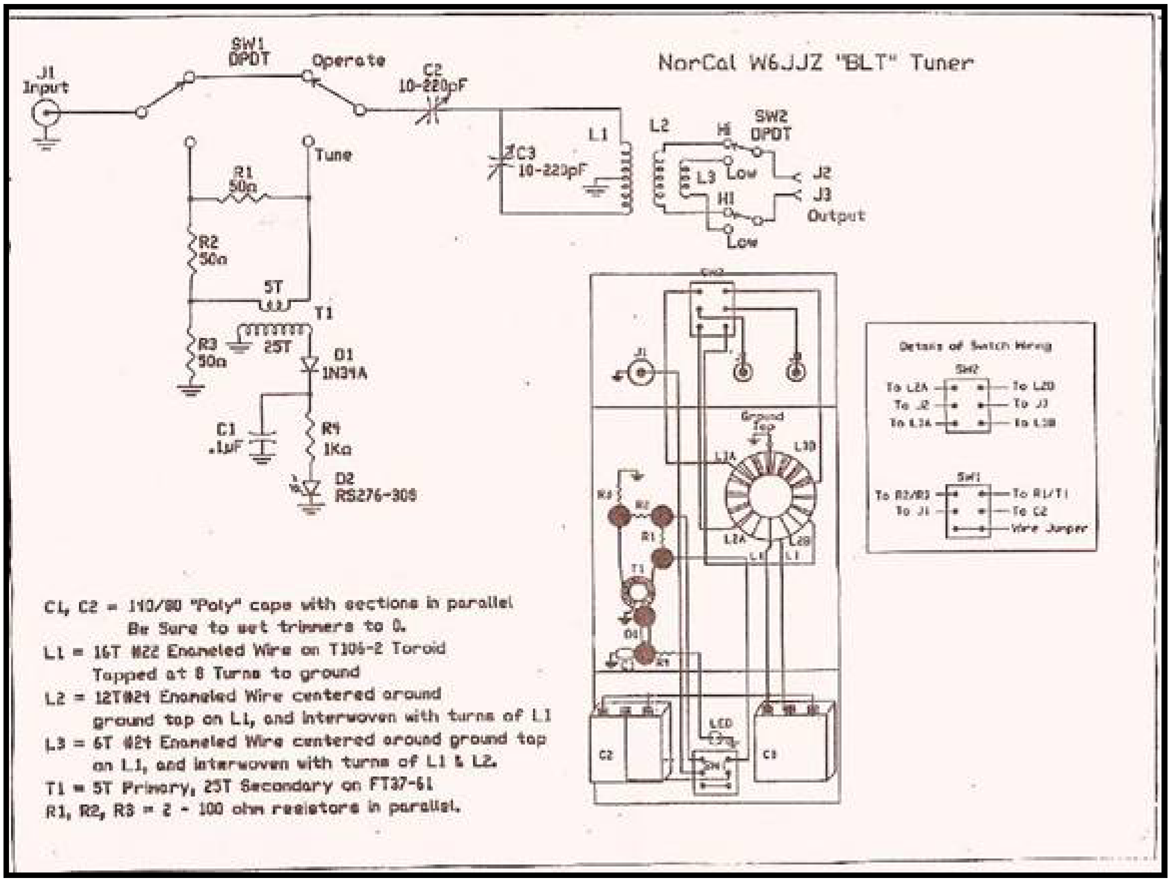
We are now ready to wire the tuner. Mount SW1, J1, J2 & J3 on the back panel. Place L1 on the bottom of the tuner where it goes using the layout as a guide. Solder the ground tab of L1 to the bottom of the case as a means of mounting L1. Then, connect the leads from L2 & L3 to J2 and J3 as shown. Then connect the leads for L1 to C3 as shown. Connect a lead from the center conductor of J1 to SW1 as shown.

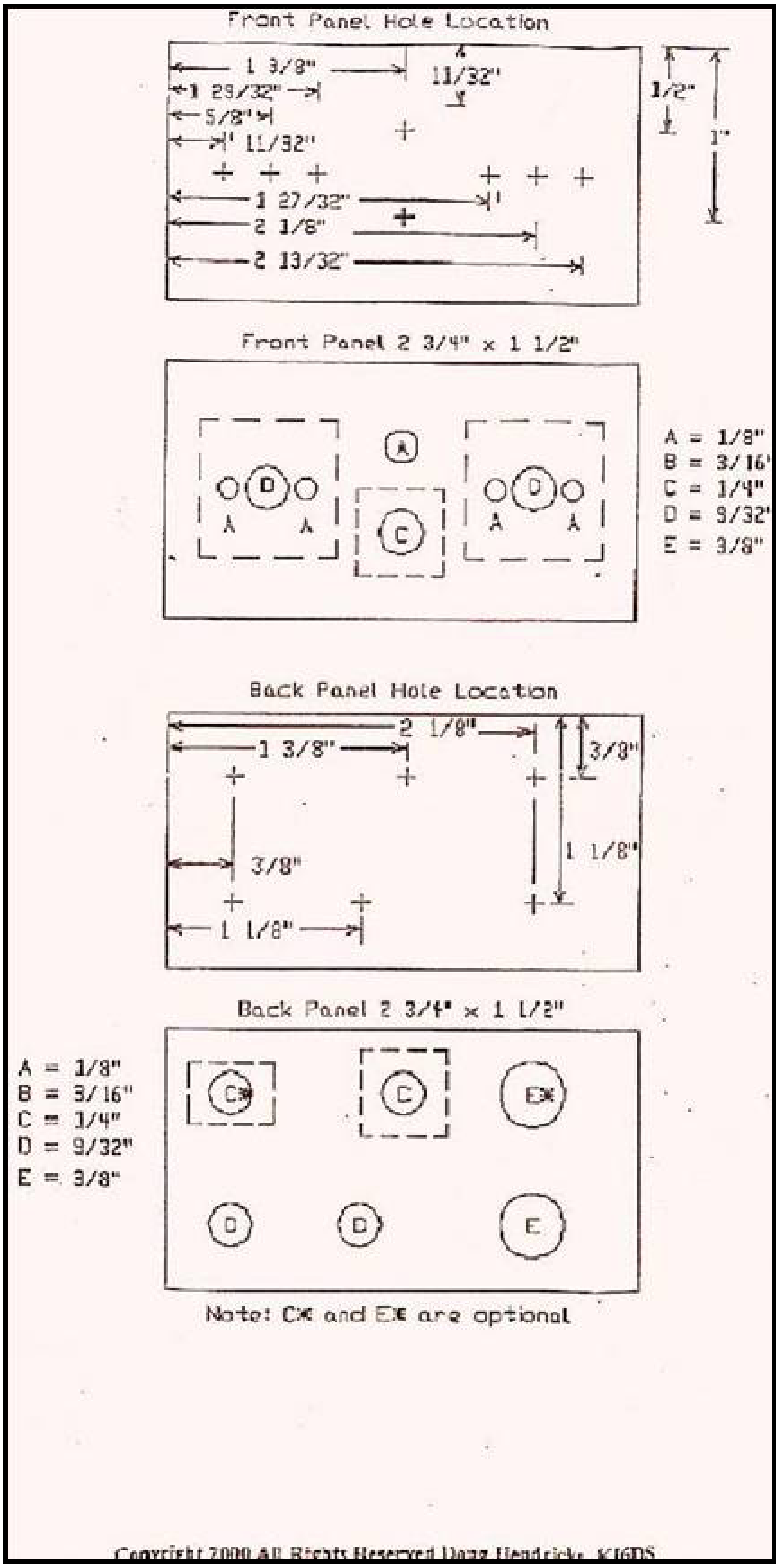
Finish wiring the connections for SW1 and SW2. Check the diagrams to make sure that you have connected all the wires. That is all there is to it. Your tuner is now finished!!

To operate the tuner, connect a balanced feedline to J2 and J3. Run coax from the BNC to your rig. Place SW1 in the Tune Position. Place SW2 in the High impedance position. Press the key or put your rig in "tune" mode, and use the two tuning knobs on the front panel to get the LED to go out, or at least dim significantly. If you can't get a match on high impedance, change to low impedance with SW2. My tuner tunes a NorCal doublet made from ribbon cable and 20 feet up in the air on all bands from 10 - 40 meters. You mileage may vary.

If you wish to use the tuner with unbalanced feedlines, i.e. coax or long wires, then you need to do the following mod. Mount a spdt toggle switch and another chassis mount BNC on the back panel. You will use the toggle switch to ground one side of the balance input connectors. Connect the center conductor of the new BNC to J2. Connect the center of the SPDT switch to J3. Connect one side of the SPDT switch to ground. To operate as a balanced tuner, switch to the unground position. To operate as an unbalanced tuner place the switch in the ground position. Simple mod. But if you are going to do it, I suggest that you do it before you build the tuner, as it is easier to do at that time. Radio Shack has a nice miniature SPDT switch and they also carry BNC chassis mount connectors.

Good luck, enjoy your tuner, and have fun on the air. Many thanks to Charlie Lofgren, W6JJZ for his invaluable assistance on this project. 72, Doug, KI6DS


NorCal W6JJZ "BLT" Tuner
J1
Input
SW1
DPDT
Operate
Tune
C2
10-220pF
C3
10-220pF
L1
L2
L3
SW2
DPDT
Hi
Low
J2
J3
Output
R1
50Ω
R2
50Ω
R3
50Ω
5T
T1
25T
D1
1N34A
C1
.1µF
R4
1KΩ
D2
RS276-309
C1, C2 = 110/80 "Poly" caps with sections in parallel
Be Sure to set trimmers to 0.
L1 = 16T #22 Enameled Wire on T106-2 Toroid
Tapped at 8 Turns to ground
L2 = 12T#24 Enameled Wire centered around
ground tap on L1, and interwoven with turns of L1
L3 = 6T #24 Enameled Wire centered around ground tap
on L1, and interwoven with turns of L1 & L2.
T1 = 5T Primary, 25T Secondary on FT37-61
R1, R2, R3 = 2 - 100 ohm resistors in parallel.
Ground Tap
LED
Details of Switch Wiring

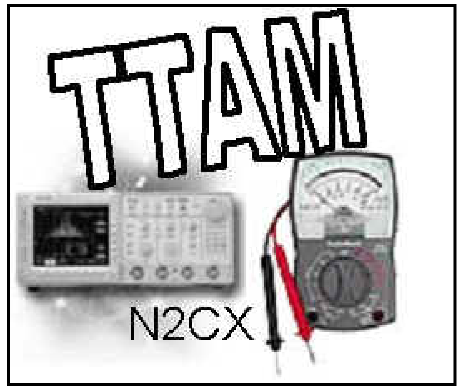

Front Panel Hole Location
1 3/8"
1 29/32"
5/8"
11/32"
11/32"
1/2"
1"
1 27/32"
2 1/8"
2 13/32"
Front Panel 2 3/4" x 1 1/2"
A = 1/8"
B = 3/16"
C = 1/4"
D = 9/32"
E = 3/8"
Back Panel Hole Location
2 1/8"
1 3/8"
3/8"
3/8"
1 1/8"
1 1/8"
Back Panel 2 3/4" x 1 1/2"
A = 1/8"
B = 3/16"
C = 1/4"
D = 9/32"
E = 3/8"
Note: C* and E* are optional

AmQRP.ORG HOMEBREWER

Joe Everhart, N2CX

TEST TOPICS

... and MORE!



"Good Enough Q"

Designed For Test

Several TTAM columns back (Ref 1, 2), I presented some of the basics needed to measure Q and examined some practical limitations in simple measurement circuits. Perusal of an electronics trade magazine recently uncovered yet another method. In this column I will describe how to connect some basic building blocks to use this technique for Q measurements and how to use almost the same method to measure Q with some test gear you may already have. The *impromptu* method doesn't always give lab-grade accuracy though most of the time it is entirely adequate for the homebrewer's needs. In this sense you might call the results a "Good Enough Q-Meter".

The article uses the fact that a series LC circuit at resonance has a total resistance of only the loss resistance of the inductor. (As was discussed in earlier Q meter installments, the capacitor usually has much smaller loss resistance than the inductor so it can be ignored.) This is illustrated in Figure 1.

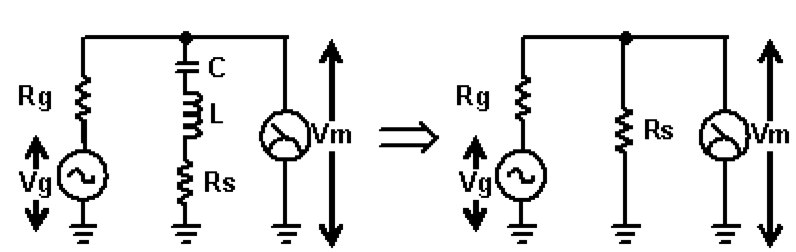


Figure 1 – At Resonance Series L and C cancel out

The tuned circuit on the left disappears when tuned to resonance leaving only the series resistance in the circuit. The voltage across the tuned circuit Vm can be calculated by the familiar voltage divider equation:

Rm = Rs * Vg/ (Rg = Rs)

where Vg is the open circuit signal generator output voltage, Rg is the generator output resistance (50 ohms for lab signal generators), Rs is the series loss resistance of the inductor and Rm is the measured output voltage at resonance. Knowing Vg and Rs you can work the equation backwards to find Rs from the measured Vm value:

Rs = Vm * Rg / (Vg – Vm)

The author used an RF voltmeter with a 50-ohm resistor so VG and Rg were reduced by half but the principle is the same. Finally Q is calculated using the reactance of the inductor XL which is calculated from the equation:

XL = 2 * pi * F * L

Then ... **Q = XL / Rs**.

A simple homebrew Q meter can be built easily using the above principles. The signal generator needs to have accurate frequency generation with a good low-distortion output and have a known output impedance. DDS signal generators described recently in the homebrew literature are an excellent choice. A good variable capacitor is needed to tune the circuit to resonance. The exact capacitance need not be known, so a readily-available capacitor is the poly-dielectric variable cap found in junked AM transistor radios. Finally a good RF voltmeter is needed. Several possibilities come to mind. First is a well-calibrated oscilloscope with sensitivity to below 100 mV. The second choice is one of the Analog Devices AD8309 or similar logarithmic detectors (www.analog.com) that have been used for wide-dynamic range field strength meters. Finally a new solution is use of a recently announced RF voltmeter adaptor called the Accuprobe. It is a detector that will allow accurate RF measurements to less than 100 mVrms when used in conjunction with a digital multimeter. The Accuprobe was announced at the June 2004 NORTEX HAMCON. More will be published on the Accuprobe in future TTAM columns.

The functions needed for this Q measuring method looked mighty familiar. A little thought revealed that they are available in the antenna analyzers many of us have recently bought. They have a high purity tunable signal source, a means of measuring RF resistance and even include the ability to measure inductance or inductive reactance! Examples are units sold by Autek and MFJ, as well as the Antenna Analyst now offered by the NJQRP using the Micro908 processor. (See www.amqrp.org/micro908.) The absolute Q measuring accuracy may not be as high as an instrument purpose built to measure Q though it is entirely usable for most homebrewing applications. If you have one of these analyzers all you need is a variable capacitor to tune the inductor and you have a "Good Enough Q-meter".

The measuring process is simple:

1. Determine the reactance of the inductor at the frequency of interest. Either measure its reactance directly or measure its inductance and use the XL formula already given.

2. Connect the inductor in series with a variable capacitor and connect this series combination to the antenna connector on the analyzer. See Figure 2 for the connections. Be sure to connect the capacitor on the ground side of the inductor and ground its rotor to minimize hand-capacitance effects during tuning.

3. Set the analyzer to the desired measurement frequency and set it to measure RF resistance.

4. Adjust the variable capacitor for minimum measured resistance. The measured value is Rs the series loss resistance of the inductor.

4a. If Rs is less than 5 ohms, measurement accuracy will be impaired. In this case it will be necessary to add a 4.7- or 5.1-ohm non-inductive resistance in series between the inductor and antenna analyzer. After adding the resistor remeasure the resistance and subtract the value of the resistor you added. The difference is Rs.

5. Calculate Q using the formula Q = XL / Rs.

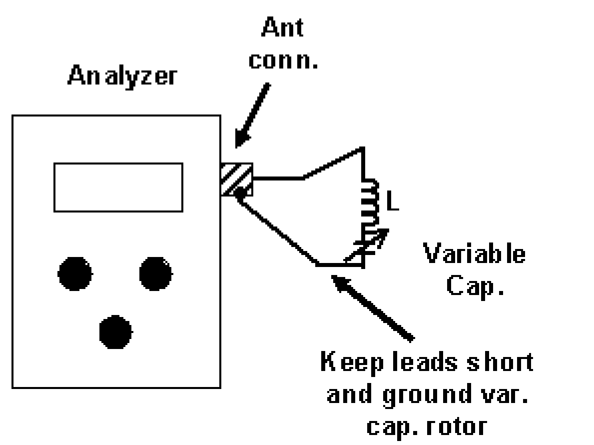


Figure 2 – Good Enough Q meter uses antenna analyzer

Coming to Terms

The terminology to be discussed this time around is "Good Enough Design". The concept was introduced in Joe's Quickie (Ref 4) in conjunction with a very simple test oscillator for use on the homebrew work bench. It was also a featured handout kit at a recent NJQRP meeting. The underlying idea was that you don't always need a multi-hundred dollar audio generator on the test bench. A simple oscillator with a fairly clean sine wave and a known output level can do 90% of the jobs you will run into for common testing.

Naturally we all want to strive for perfection but it's not always in our best interest. We have to carefully think about what we are trying to do and what's needed to do the job. Our QRP philosophy is an example of this idea. In most cases when the primary goal is to merely communicate and to enjoy ourselves while doing so we really don't need oodles of power.

We are often faced with constraints when it comes to what test equipment we want to buy or build. We never have enough money to equip our workbench with brand new HP (oops, now Agilent) or Tektronix gear, we don't have enough time to build complicated test gear when the goal is to lash up something to test for something else we're building. We don't have the necessary gear to calibrate fancy stuff even if we do build it and we are faced with the knowledge that much of modern electronics technology is very difficult to incorporate without long experience and training. The bottom line is that we need to very carefully determine exactly what we want to do with our test gear, how much time, effort and money we can expend on it and how much use we will have for it in the end. In short we have to decide what is "Good Enough" to get the job done.

In truth those in the electronics industry are more often than not faced with the same issues, but "Good Enough" may be a tad more elaborate than in the home workshop. Most of the time industrial jobs are constrained by either a specification or some other agreed-upon guidelines for a given project. Much the same as hobbyists, design engineers want to strive for the utmost perfection in whatever they do. The reality is that managers keep reminding them that perfection is nice but their guidance has to be "what does the specification call for?" Anything beyond that is unnecessary and a waste of time and money.

Stimulus and Response

At the 2004 Atlanticon QRP conference held in Baltimore, Maryland, I discussed some recommendations for component measuring gear. Interestingly, I've since been asked for recommendations from several folks who could not attend the conference.

The recommendations I made were simple. First of all we should all own a capable antenna analyzer. Several good ones are made by MFJ (Ref 5) and Autek (Ref 6) and as mentioned above a new homebrew one is now available from AmQRP. Basically you want an instrument that will measure SWR and antenna impedance, and preferably also be able to measure the resistive and reactive antenna impedance values. Having such an instrument lets you measure lots of other things in the shack besides antennas (like Q as we saw above).

Second, if you want to do more precise measurements you might want to consider some specialized instruments. For L and C measurements you might consider the AADE LC Meter II (Ref 7) or the less expensive but more homebrew Elsie Kit offered by NJQRP. Also in this category is the unique Atlas LCR Passive Component Analyzer that automatically identifies R, C and L components and provides a digital readout (Ref 8).

A final handy-to-have item is the Atlas DCA Semiconductor Component Analyzer that makes semiconductor testing very easy. It has three leads to connect to an unknown semi device. When these are connected (or two of them for diodes) and the device is turned on, it automatically determines which type of device you have, what the device leads are and even reads out its DC electrical characteristics. It handles ordinary diodes, LED, NPN and PNP bipolar transistors, and both P- and N- channel MOSFETs and JFETs. I have one and though I could live without it, the automatic operation makes checking unknown or suspected semiconductor devices an absolute joy.

One of these days I may just reverse-engineer that puppy!

References:

1. Test Topics and More #14, ARCI QRP Quarterly, January, 2003

2. Test Topics and More #15, ARCI QRP Quarterly, April, 2003

3. Alan Victor, EDN, Feb 21, 2002 "Method simplifies testing high-Q devices"

4. Joe's Quickie # 47, WA8MCQ Technical Topics column QRP Quarterly, Oct 2003

5. http://www.autekresearch.com/

6. http://www.mfjenterprises.com/ products.php?catid=49

7. http://www.aade.com/lcmeter .htm

8. www.peakelec.co.uk

9. Design Ideas - available on-line at: www.reed-electronics.com/ednmag/contents/images/22102di.pdf

Joe Everhart, N2CX may be contacted at 214 New Jersey Road, Brooklawn, New Jersey 08030. Joe may also be reached by e-mail at n2cx@amqrp.org

AmQRP.ORG HOMEBREWER

Rich Arland, K7SZ

Coming Back to QRP

It's been several issues since I have had the good fortune to be featured in ***The Homebrewer*** magazine. By way of explanation let me first publicly offer my apologies the readers of this publication. Shortly after wrapping up the 2nd edition of ***The ARRL's Low Power Communication, The Art and Science of QRP,*** I suffered a "ham radio burnout" of sorts. I had no desire to pursue the ham radio hobby in general and QRP in particular. Hey, it happens, OK? Although I had assured George that I would be sending him regular installments of my quarterly column for ***The Homebrewer*** magazine, I begged off at the last moment, leaving him with some column space to fill and no author available to do the job. To put it bluntly: I let down George N2APB and the American QRP Club in spades. In addition, I was totally inactive on the bands starting at the middle of January 2004.

Flash forward to September of 2004: My copy of the QRP ARCI Quarterly magazine had arrived and I started half-heartedly reading through the pages of QRP information. Little by little I began to feel the "QRP Twitch", deep within my soul once again! In about a week, I found that I was back on the air, albeit not all that frequently at first, using QRP CW to work some of the DX stations I saw on the ARRL DX Bulletin. By the end of 2004, I was putting in some serious air time and had constructed an Elecraft KX-1 transceiver kit with all the options. Additionally, I had acquired an ICOM IC-202 2M SSB/CW portable (?) rig and was deeply involved in plotting how to put together a VHF weak signal station without raising the ire of my wife, "The Beautiful and Talented Patricia", since she wanted her bathroom finished (we've only been remodeling that portion of the house for 12 years!). To sum it up: "I'm Baaaaaaaaacccck!"

With humble heart I approached George, begging his forgiveness for my inability to keep up with my end of the bargain. He graciously welcomed me back into the QRP Fold and immediately signed me up to emcee Atlanticon 2005! What a come back!

This brings me to the topic of this column: Why we all "come back to QRP".

In 2005 I celebrate my 40th year as a QRPer. I joined the ***QRP ARCI*** in 1965, while still in college. In those days I, like most everyone my age, felt invincible and wanted to "change the world". QRP seemed like a great way to approach the ham radio hobby and accomplish some "RF Ecology" at the same time. During the intervening years I found that not everyone shared my rose-colored-glasses approach to the world of ham radio. What I experienced for many years was an almost total disdain by main stream radio amateurs, especially the "high power boys", toward those of us who dared use QRP power levels to DX, contest and rag chew. “Life is too short for QRP” became the mantra of those high power ops that looked down their collective noses at us QRPers. After all, how could we successfully contest or accomplish DXCC, let alone 5 band DXCC, with only 5 watts or less when they (the QRO crowd) took years to do the same thing using 1000 to 1500 watts (or more, sometimes much, ***much*** more…depending upon who was telling tales out of school after waaaaay too much alcoholic libation).

My thanks to people like Ade Weiss, W0RSP (then K8EEG), Roy Lewallen, W7ZOI, Doug DeMaw, W1FB (then W1CER), and countless others who shared their vision, expertise and, of course, their homebrewing talents with the rest of us mere mortals. These people were all my personal QRP heroes when I was struggling to make contacts using only 5 watts. They provided me with information and most of all, hope, that what I was attempting to do was right, proper and would actually work!

During my 20 years of active duty with the U.S. Air Force, I was fortunate to be stationed overseas for almost 15 years. This allowed me to operate QRP from Japan (twice) the Azores , England , Wales , Scotland , Germany , and many places around the United States . During TDYs you could find me outside the VAQ stringing up some kind of HF antenna to hook my HW-7 or HW-8 to in order to pursue my ham radio hobby. There are a lot of houses I’ve lived in around the world that have the remnants of attic antennas, verticals, end-fed wires, etc, hanging around long after I had left. All in the name of QRP!

My most prosperous tour (ham radio wise) was my five plus years in the United Kingdom , stationed at RAF Mildenhall, East Anglia ….. Suffolk , to be precise. I vividly remember running a piece of wire out of my rear bedroom window to the chain link fence behind my off-base house in Bury St. Edmunds, and hooking it up, via a small tuner, to my Ten-Tec PM-2B. That fateful night I managed to work a guy by the name of Colin Turner, G3VTT, near Dover . He sent me a QSL card with some obscure information on the reverse side about the ***G-QRP Club*** and some guy named Dobbs. It turns out this guy was a “reverend” in the C of E (Church of England…Episcopal to you Colonists). It seems that the Right Reverend Dobbs, G3RJV, had formed this group of QRPers in England and if you collected several QSLs from the various members, turned them over and taped them together, you ended up with a schematic of a simple transistor transceiver that you could build and play the QRP game with. Hmmmmm, I gotta meet this guy.

And meet him I did! As a matter of fact, my wife, “The Beautiful and Talented Patricia”, coordinated a surprise 37th birthday party for me at the QTH of the Rt. Rev. Dobbs, and his wonderful wife, Jo. In attendance were about 20 ***G-QRPers***. Of particular note was George Burt, GM3OXX and Ronnie, GM4JJG, both of whom traveled from Scotland bringing with them some prize winning haggis. For those of you who’ve never had the pleasure of tasting this Scottish delicacy let me assure you that once you make it past the smell, you are in for a real taste treat! Seriously, it was ***REALLY*** good! Unfortunately, I was unable to recite Bobby Burns’ “Ode to the Haggis”, which (being a fellow Scot myself) was a bit embarrassing, to say the least. However, Ronnie rose to the occasion and performed the “honours” and soon we all partook of the unusual taste of Scottish haggis. With single malt whiskey flowing in abundance, a great time was had by all in attendance. I look back on my time in the UK , hanging around the ***G-QRP*** crowd, as the most informative and productive time I’ve ever spent as a ham radio operator. These fellows liked me, despite my being a “colonial”, and they took me under their collective wing and nurtured my homebewing talents (such as they were at the time). This resulted in my becoming a much more skilled QRP operator and an addicted homebrewer.

Recently I picked up a copy of ***The QRP Handbook*** written by G3RJV and published by the ***RSGB***. Looking over pages of this outstanding book, I was instantly transported back in time about 20 years to my days in the United Kingdom and my association with the ***G-QRP Club***. This work is a collection of articles from the G-QRP Club newsletter, ***SPRAT*** (Small Powered Radio Amateur Transmissions). As I thumbed through the contents, I ran across circuits like the “Sudden Receiver”, one of which I still use on 160 today. Then there was the “Stockton Wattmeter”, a four port RF bridge circuit that was accurate down to the milliwatt power levels. Mine still finds regular use in the K7SZ shack, even though I built it almost 20 years ago! GM3OXX’s “OXO” transmitter was a hoot to build and operate, and mine is on the shelf above the workbench, awaiting my use. What a great collection of circuits and memories. These guys know how to play the QRP game!

Upon returning to the States, I was frustrated and disappointed at the lack of homebrewing along with the absence of club projects within the ***QRP ARCI***. Stateside QRPers do not participate in the hobby the way the UK QRPers do. Mostly this is due to the excessively high costs of commercial gear in Europe, and the wonderful Value Added Tax (VAT) that ups the cost another 15% (at least it was 15% when I was over there 20 years ago!). The Brits (and the majority of European QRPers) tend to homebrew their radio equipment primarily due to cost, but also with an eye toward the pride in accomplishment derived from designing and building one’s own station gear. And, if you want to see some really, ***really*** beautiful homebrew equipment, look no further than the utterly show-stopping creations from George Burt, GM3OXX! When it comes to fantastic homebrew gear, George has no peers. He ***is*** the king!

Conversely, stateside hams would rather buy than build, which was a major culture shock stumbling block when I returned to the U.S. After a couple of terms on the ***QRP ARCI*** Board of Directors, I became disgruntled and withdrew for a couple of years, all the while longing for the “glory days” of my time in the UK, where homebrewing was king and the exchange of information was paramount at QRP gatherings.

When Doug Hendricks, KI6DS, started the ***Northern California QRP Club*** (***NorCa***l), I was ecstatic. The ***NorCal*** goal was to further the art of homebrewing, and to start the ball rolling, Wayne Burdick, N6KR, designed the now-famous NorCal 40, a 40 meter CW transceiver (currently being sold as the NC-40A by ***Wilderness Radio***). This design was simple, easy to duplicate, worked the first time, even when built by relatively inexperienced builders, and produced many hours of enjoyment on 40 meters for the enterprising QRPer. This design was so successful that it became an undergraduate college level project at Cal Tech. The NorCal 40 was followed by the Sierra, the SST, and then along came Eric Swartz, WA6HHQ, who teamed up with Wayne Burdick and ***Elecraft*** was born which gave us the K2 , K1, KX-1 transceivers. All proving that homebrewing was alive and well in the U.S. Essentially, it all started with the NorCal 40. Wow, talk about changing the face of ham radio! I was in heaven!

Then along came this dude, George Heron, N2APB, and his sidekick Joe Everhart, N2CX, and the ***New Jersey QRP Club***. Finally: a club on the east coast, close to my home, that followed the NorCal lead and started pumping out QRP kits and accessories faster than a rabbit in heat slings a litter! My prayers had been answered!

All this history brings me to the present. Like many of you I have had my “ins and outs” of the ham radio hobby over the years. I’d leave the QRP fold to explore another facet of the ham radio hobby, only to return sooner than later to QRP. Why? Was it the challenge of using 5 watts or less RF output to work the world? OR, was it the chance to build innovative transceivers, accessories, and kits that rivaled the Offshore Empire offerings? Possibly it was the chance to thumb my nose at the QRO ops who, for so long, looked at us QRPers as quirks of nature, and not highly skilled radio operators who could run circles around them contesting, DXing or just plain having fun with the radio hobby.

Actually, upon long term reflection I came to the conclusion that it is the people within the QRP hobby that draws me back into QRP each time I stray. It’s the George Dobbs, the Collin Turners, the George Burts, the Ian Keysers, the Dave Bensons, the Eric Swartzs, the Wayne Burdicks, the George Herons, the Joe Everharts, the Tom McCuens, the Bob Dyers, the Fran Slavinskis, the Paul Strouds, the Ed Breneisers, the Ron Politykas: It’s all the QRPers that I have met and that have shaped my persona. It’s the folks who take the time, make the effort, and volunteer their services to help the QRP hobby grow that bring me back. My good friend and SWL mentor, Dr. Harold, “Dr. DX” Cones, of the ***Old Dominion DXers*** in Tidewater Virginia has a saying: “Leave the hobby a bit better off than you found it”. That is what these QRPers do. That’s what we all do when we take the time to mentor a new comer to the hobby or design and share a circuit, or help someone debug a problem radio, or volunteer our services to publish a newsletter or help manage a QRP club.

While we ham radio operators do build and we do talk (communicate) our biggest gift is our ability to help grow the hobby. That is what brings me back; the people.

That’s a wrap for this time, gang. Over the upcoming issues I will introduce you to “The Argonaut 509 from Hell”, the Wyoming Valley QRP Commandos, and we’ll discuss a variety of topics to help you expand your QRP horizons. Until next time: vy 73.

Rich K7SZ

PS: If you have any of the following ICOM gear and want to find it a “good home” please contact me at: richard.arland@verizon.net and we’ll get together on a deal:

IC-502 6M SSB/CW VHF QRP Transceiver

IC-402 70CMs SSB/CW UHF QRP Transceiver

IC-202S 2M SSB/CW VHF QRP Transceiver

IC-215 2M FM Crystal Controlled Transceiver

IC-515 6M FM Crystal Controlled Transceiver

Rich Arland, K7SZ may be reached by mail at 25 Amherst Avenue, Wilkes Barre, PA 18702, or by email at Richard.Arland@verizon.net

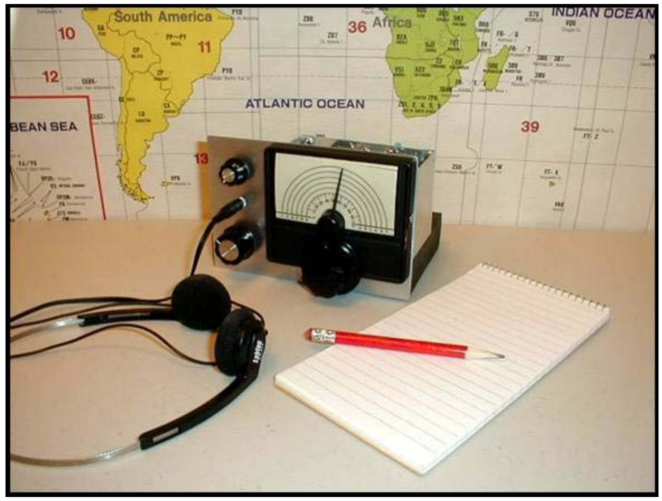
Editor's Note: As with most of us, we never really leave QRP or even ham radio, but sometimes take sabbaticals for reasons of family, work or physical/mental health. K7SZ has been a long-time contributor to the QRP community and to the *state-of-the-art* as we know it today. He is a member of the QRP Hall of Fame and has given of himself more than anyone I know ... and we are very proud to have Rich back with us on the HOMEBREWER and Atlanticon.

AmQRP.ORG HOMEBREWER

Richard Fisher, KI6SN

QRP Operating

"Joys of the Classic Regen Receiver"



From **Bill Jones, KD7S,** of Sanger, California: "Many years ago I was transferred to a strange city to begin a new job. It would be a month or so before I could get my wife and children relocated with me. I spent my nights in a sleazy hotel room with nothing for entertainment but a single tube, regenerative shortwave receiver. That little receiver preserved my sanity.

"The original receiver has since been lost in the shuffle, but from time to time I feel compelled to build a new one. I can't begin to describe the warm, comfortable feeling I get from listening to voices from half way around using a radio I built myself.

"Just before Christmas 2003, I ran across a brand new James Millen vernier dial in a forgotten box of parts. I remember buying two of these dials at a hamfest nearly 20 years earlier for a dollar each. Holding the dial in my hand and feeling the velvet smoothness of mechanism flooded my mind with memories of my old regen receiver. It was time to heat up the soldering iron.

"I rummaged through a desk drawer and found a PC board from FAR Circuits for one of Charles Kitchin's designs. An excellent description of this receiver can be found on Ian Purdie's web site: http://www.electronics-tutorials.com/receivers/regen-radio-receiver.htm

"I must admit that I took some liberties with Mr. Kitchin's design. For example, I was never really comfortable with the main coil design. It was just too big and cumbersome to suit me. Taking a page from the history books, I used an old octal tube base as a coil form. A phoenolic tube socket was mounted on the rear of the chassis to accommodate plug-in coils. Once the correct number of turns was found I put a layer of black plastic electrical tape over the wires to keep them in place. Like they say on television, 'it works for me.'

"I haven't gotten around to labeling the controls yet. The ON / OFF switch is part of the volume control in the upper left hand corner. The regeneration control is at the lower left. The rear view shows how the coil is positioned just in front of the PC board.

"The regeneration capacitor is behind the 9-volt battery. The receiver tunes from 9.3 to 13.5 MHz with a 10-40 pF main tuning capacitor. However, by plugging a 100 pF capacitor into the tube socket where the main tank coil is connected I can also tune from 5.04 to 6.50 MHz.

"The front panel and brackets that hold the PC board are made of .032" soft aluminum sheet. The chassis was fabricated from 1/8" ABS plastic.

"The performance of Mr. Kitchin's receiver is outstanding. The 31 and 49 meter bands are filled with signals during the evening hours here on the west coast. During the daytime I can hear numerous stations on the 25 and 21 meter bands. The regeneration is smooth and predictable and there is enough audio gain to drive a small speaker on the stronger stations. Best of all, I still get that warm and comfortable feeling listening to far away voices on a receiver I built myself."

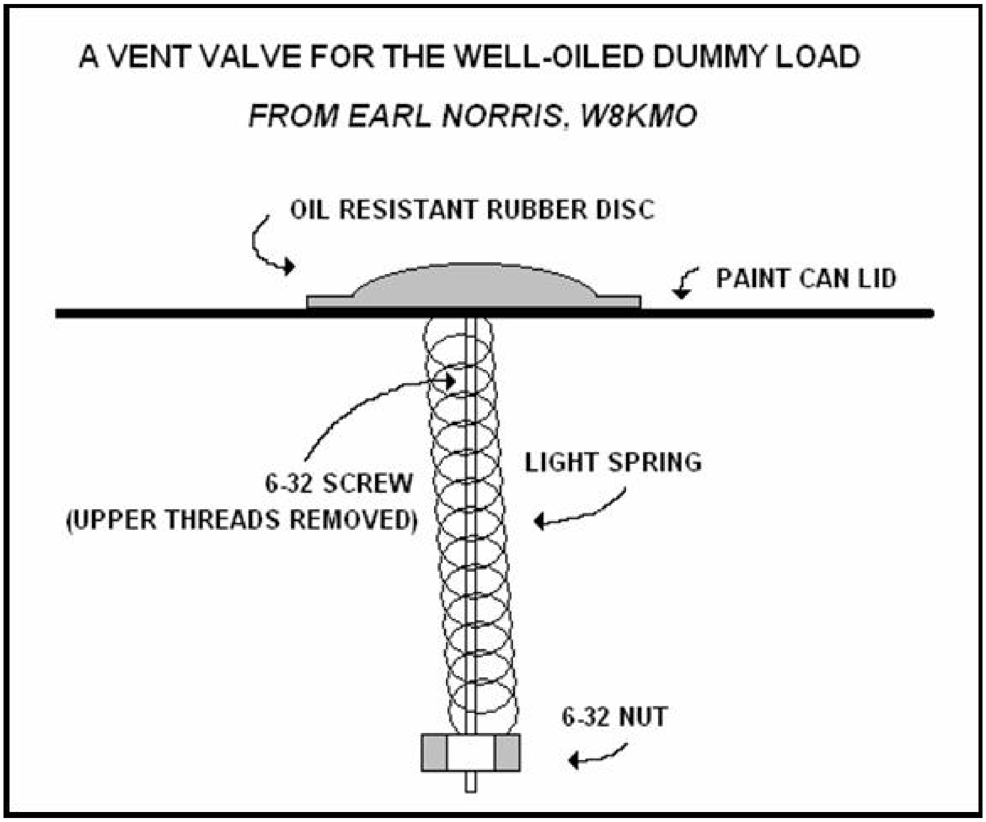
* * * * * * *

MORE ON A WELL-OILED QRP DUMMY LOAD

Earl Norris, W8KMO, writes to say he enjoyed Homebrewer No. 2's piece on dummy loads. "I think it was great and needed.

"As I was reading the article I had a couple of thoughts I decided to share with you. As for a nice can to hold the cooling oil, some paint stores carry new, unused paint cans in gallon and quart sizes. Some might have pint cans also.

"I have attached a drawing of a vent valve for the lid of a metal can. The 6-32 screw would need to have most of the threads removed to keep it from catching on the edge of the hole in the can.



"This pretty much like the vent valve used on the Heathkit Cantenna. I do not think one should create heat in a sealed can of oil.

"Second idea: When calculating the value of resistors in parallel, if all the resistors are of the same value just divide the value of one resistor by the total number of resistors. If the resistors are of different value it becomes a pain. I taught electronics school for the Navy and many of our students had trouble with dissimilar resistor values.

"Again, thanks for a great article. I hope many folks read it and take the advice to heart."

* * * * * * *

ALL IN THE FAMILY – A QRPer's TALE

Michael F. Danchi, KE4TSA, writes, "For those who don't know, I'm a college kid studying music in Rochester, New York at the Eastman School of Music.

"I have a family in the area – the Browns – that has unofficially adopted me while I'm at school and away from my family in North Carolina. Five of them are licensed, and their radio (TS-850 with power supply and internal auto-tuner – a gift from their father / grandfather) has been sitting in the basement unused, due to a combination of lack of time / energy / experience.

"I found out about it and promised the kids that I'd help them get it up and running.

"Recently, Keith Hibbert, WB2VUO, ARRL Technical Coordinator for Western New York, came over to their place and helped me make good on my promise. I've never owned equipment like this before, so I really didn't know how to get the station and antenna set up right.

"Keith spent the entire day (1000 to 2000 EST) helping us get everything set up and running – which included a couple trips to the hardware store for supplies we didn't have.

"The operators include Mikel, KC2EMQ, who is the dad; Julie, KC2FER, the mom; Sarah, KC2FES, sister; and brothers Jimmy, KC2EMS, and Timmy, KC2FEQ.

"The station is now fully functional on all bands (160-10 meters) and contacts have been made from coast to coast. Five hams now have access to high quality HF operation. Make that six, if you include me.

"I hope to encourage these guys / girls to take part in serious CW operation and eventually QRP activity.

"If anybody is interested in helping me, listen for Jimmy, KC2EMS, who is the primary station operator. He'll probably be on 40 and 20 meter SSB (General class bands) for now. A little encouragement – especially to try CW and QRP – would be MUCH appreciated.

"People like Keith make amateur radio what it should be. Code or no-code, if we're going to be good operators we need people like him to get us on the air, help us learn the ropes, and teach us how to operate responsibly and effectively.

"So, thank you, Keith! Keep up the good work."

* * * * * * *

NINE YEARS TO COMPLETE A RIG – CAN YOU BEAT THAT?

Andy Palm, N1KSN, writes from Menasha, Wisconsin, that "in 1995, just before I went QRT for seven years upon our move to Wisconsin, I started building an SW80 from Small Wonder Labs, mostly because the sunspot cycle was in decline.

"I finished the PC board and had gathered jacks and pots and a cast aluminum case, but stopped before getting it together.

"Fast forward to Spring of 2002, when the FT-817 and MP-1 lured me back on the air. I eventually dug into my boxes of ham stuff and found the unfinished kit.

"It took a while, but I found the manual and completed the assembly. However, at that time I was still 'alignment challenged,' so I sent the rig off to SWL's repair guy Steve, together with a request to expand the VFO range – a capacitor change.

"Both Dave and Steve were a great help, and I got the rig back aligned, with the requested modification, coax added between the board and RF BNC connector, and with a few 'touch-ups' of questionable solder joints – seven years after I originally bought the rig. Is that great service, or what? The fee was modest, too.

"Unfortunately, as Steve pointed out, although the tuning range was wider, the power now dropped off significantly at either end of the tuning range because of the final bandpass filter.

"Somewhat disappointed, I put the rig away. I didn't even have an antenna for 80 meters. A year and a half later I tried selling it at a hamfest. No luck.

"Again it went into storage, until early 2004 when, suffering from serious solder fumes withdrawal, I got it out again.

"It was easy to replace the capacitor controlling the VFO range to a value slightly smaller than the original specification, and my alignment skills have improved considerably after building several rigs in the last couple of years.

"The result? A great little rig for 80 meters with 1.5 watts across the entire tuning range of 3.539 to 3.565.

"After nine years I finally got it on the air and had some great contacts on night-time 80 meters, a band I've unfortunately neglected too long. I didn't even miss having RIT.

"I'm very happy that I finally got this transceiver kit in good operating condition, but it sure had to wait a long time.

"Do you have a similar tale? Can you beat a build time of nine years for one of your kits?"

* * * * * * *

A RANDOM ACT OF QRP KINDNESS

Joe Trombolino, W2KJ, writes from Hampstead, North Carolina that "every so often we see random acts of kindness taking place on the Internet Mail Group QRP-L and such an act occurred just a few days ago.

"I had posted a ‘want’ for an RH-20 transceiver kit. Having just completed an RH-40, I thought it would make a nice pair.

"I received a response from Jim Johns, KAØIQT, of Trophy Club, TX, who said he thought he had one laying around somewhere in his shack. He said he would look for it. ‘By the way,’ Jim said, ‘what is your shipping address?’

"I sent him my address and asked how much he wanted for the kit. I hadn’t received a reply for several days and wondered if perhaps he never found the kit.

"Much to my surprise I found a package on my front stoop from Jim. It was the kit with a letter enclosed that pretty much sums up the spirit of amateur radio.

"The note said essentially that if I accepted the radio free of charge when I was done playing with it, I should pass it on – free of charge – and ask the same of the person to whom it was given.

"I was totally flabbergasted. This was such a random act of kindness.

"I guess these things happen occasionally but I have seen this happen more on the QRP-L than anywhere else.

"What a great way to get others involved in radio if they don’t have a rig. I’m thinking about youngsters here who may get licensed but don’t have the bucks for an HF rig.

"I accepted Jim’s terms. I also have another idea on how to put the rig to good use. I am thinking of using it as a ‘QRP Mentoring Device.’ That is: loan it out to local HF’ers who have never tried out a QRP rig.

"Perhaps I could spark some interest in QRP and homebrewing / building at the same time.

"I owe a very big THANKS to KAØIQT for his efforts in furthering the hobby and getting new folks into it.

"I hope I can put the RH-20 to good use Jim. I hope Jim isn’t offended by making his act of kindness public, but I feel folks like him deserve recognition among their peers for good deeds done."

* * * * * * *

Do you have QRP Operating stories to share? Drop an e-mail to: KI6SN@aol.com

Via U.S. Postal Service, write: Richard Fisher, KI6SN, 1940 Wetherly Way, Riverside, CA 92506

AmQRP.ORG HOMEBREWER

James Bannett, KA5DVS

Radio To Go

"QRP Events"

I would like to dedicate this column to the memory of Jim Cates, WA6GER. Jim passed away recently and he will be missed. The NorCal QRP Club meetings will never be quite the same without a greeting from Jim. He was always there with a smile and a quick comment. For a while when I first moved to California, Jim would call me Bob when he saw me. I didn't correct him immediately and he soon learned that we shared the same first name. He continued to say "hello Bob" and grin when he saw me at the meetings. It became a sort of inside joke between us. I will especially miss being called "Bob".

Jim Cates, a gentleman and a true individual; he will be missed.

* * * * * * *

Radio to Go: QRP Events

As I write this section, I am flying at about 33,000 feet somewhere over the midwest on my way to Baltimore. I am making the trek from California to attend one of my favorite QRP events: Atlanticon.

The NJQRP group, led by George Heron and Joe Everhart, started Atlanticon several years ago. It was modeled after the already-successful Pacificon event sponsored by Norcal. QRPpacificon was begun as a west coast get together for QRPers and continues to be an annual success. As a transplanted NJQRPer, I look forward to seeing the New Jersey gang at Atlanticon and renewing old friendships.

Now that Norcal and NJQRP have joined forces to form AMQRP, events such as these are becoming more commonplace. Today there is Pacificon, Atlanticon, Ozarkcon, Lobstercon, and others scattered around the country. Let's not forget the first of these QRP forums ever to be held: Four Days in May, sponsored by QRP ARCI. The net result is that QRPers from most of the country have some type of event somewhere within driving distance of their home. Having attended Pacificon for the past six years or so, and Atlanticon for the past two years, I highly recommend attending at least one of the events near you.

There is nothing quite like the energy of the QRPers that these events attract. The atmosphere is difficult to capture in words but must be fully experienced. Between the building contests, operating events, seminars and vendors, there is a lot to see and do. Not to mention that many of these events are held in conjunction with major ham fests, thus adding to the fun.

When attending these events, the only thing to worry about is having too much fun and having enough space in the luggage for all the goodies.

What goes on at the QRP events?

Things typically kick off on Friday evening as people begin to arrive, check in to the hotel and meet and greet new and old friends. An informal dinner is typically the order of the day usually at a nearby restaurant. At Atlanticon, a group gathers at a nearby restaurant for a seafood dinner. Crab cakes and a brewed beverage are the order of the evening. Sitting next to George Heron and across from Dave Benson, I really felt immersed in QRP. A crowd is a given and it is difficult to hear over the rush of excited voices. Ideas are exchanged, stories swapped, kit building and designs discussed as well as many other topics too numerous to mention. Smiles and animated discussions abound and are only broken up by the necessity of returning to the conference center for the evening events.

Next, a return to the hotel and a quick trip to the room to gather stuff and head down for the evening events. At Atlanticon, the Friday evening events were kicked off with an introduction by George Heron that led into a great presentation by Wayne Burdick of Elecraft. Several vendors had a selection of various items to show or sell. Keys, antennas and a wide range of kits are available on the tables – more so this year than at any time previously. Other folks bring along their latest construction project for show and tell. The photo above is one of the typical displays of attendee's projects and homebrew works of art.

An excited atmosphere is evident and the evening passes all too quickly with excited and animated discussions continuing until almost midnight. The staff of NJQRP worked late into the night making sure everything was ready for the next day.

Saturday dawned early, especially for those of us still on Pacific time. A quick trip to the swap meet in a rain shower to check out the event then back to the hotel for the days series of excellent presentations.

At the end of the day, having heard great presentations on field operating, antenna theory and digital design, one finds himself in "information overload" – but what a feeling!

The evening wrapped up in the hotel bar over a brewed beverage or two with more discussions.

About 12:30, I finally made my way up to my room and to sleep.

Sunday has no planned events so another early trip to the swap meet was in order to look for bargains. I found lots of stuff, including a few boat anchor type rigs that are my weakness. However, the necessity of fitting them into my luggage prevented any new acquisitions.

I did end up with an ICOM 6-meter rig, something I have wanted to find for a while as well as a few connectors and various odds and ends. One of the surplus dealers had great small tripods for $10. These are perfect as a support for a portable antenna or even a camera.

Part 2

I am now on another flight on my way to Joplin for Ozarkcon. The Four State QRP Group hosts the event, with the sponsorship of the AmQRP. I am attending this year and taking along a few PAC-12 kits.

It is interesting to compare Atlanticon with Ozarkcon. The events are roughly similar in size and scope but each has distinct differences in geography and population. It will be good to see some old and new friends as well. As I was born and raised in northwest Arkansas, going to Joplin is a bit like returning home.

Ozarkcon is put on by the 4SQRP group with organizers Gene Sailsbury and Joe Porter. A group of dedicated local members also help organize and run the event.

Gene Sailsbury and Joe Porter, the organizers of Orzarkcon 2004, point to map showing originations of all attendees.

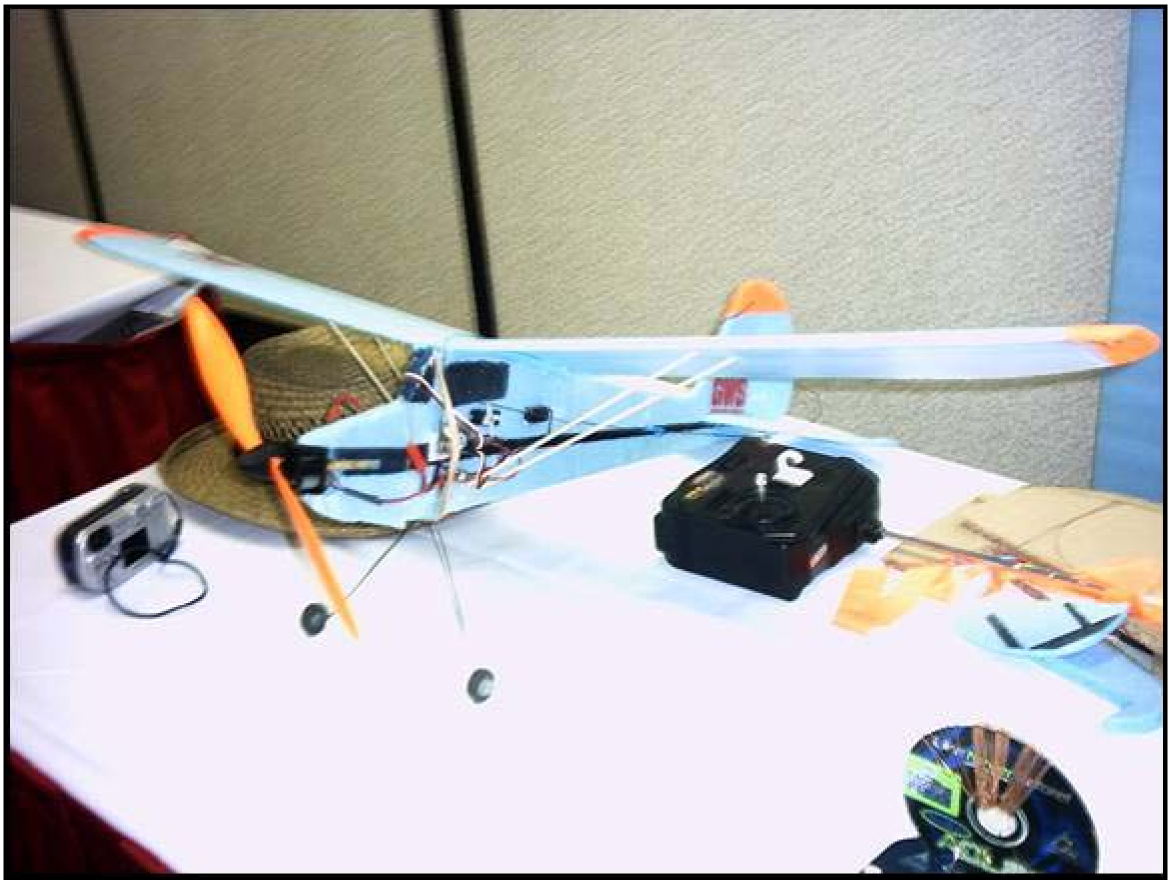
Having attended both Pacificon and Atlanticon, I can say without reservation that Ozarkcon ranks up there with the best of them. Organization is similar to the previously mentioned events but with some notable local flavor. For one, they scheduled the event around the hamfest with four speakers for the day on Saturday. This gave time for a quick trip to the nearby hamfest both before the talks commenced in the morning as well as during a long lunch break. As a hamfest junkie, I took advantage of both opportunities and came home with a few small goodies.

Ozarkcon began Friday evening with a dinner at local steakhouse. The buffet there was excellent with a wide range of choices. The meeting room was packed by the time I arrived a bit late and finally found a chair. I was sitting next to Bob Dyer of Wilderness Radio but did not recognize him, as we had not met in person. We ended up talking a bit and I learned about his business. Bob produces and sells the Norcal 40A, the Sierra, as well as other kits under the Wilderness radio name. He recently moved the business to Joplin from California and therefore was "local" to the event. After dinner, we all made our way back to the hotel and the reserved gathering room. By the time I got there, a crowd was already milling around looking at various tables of QRP toys. A building contest was held and there were some amazing entries. I am always impressed with the ingenuity and artistry of homebrewers and the quality of the construction work seen at these events.

Examples of the rigs on display at Ozarkcon 2004.

I would like to commend the Ozarkcon crew. For a group new to this, they did a fantastic job. I especially liked the speaker lineup printed on the back of the badge. This was a great idea. Also nice was the US map with pins for the hometown of each attendee. Was really a nice way to show where everyone came from in getting to Ozarkcon.

With the proliferation of such events, almost everyone in the US will have a QRP weekend within driving distance and I strongly encourage you to support these events by attending and participating. Bring along your latest construction project or idea and share with the group. It doesn't have to be pretty; you will find interest from many regardless of what you bring. At Ozarkcon, there was even a builder who brought along his home build RC airplane constructed from foam sheets. It gathered considerable interest and discussion.



You'll find all kinds of "gear" on display at these QRP events!

So, until next time, my best wishes to you all. I have a busy summer and fall travel schedule and will be operating from various locations as time permits. As always, I will be taking my radio to go.

James Bennett, KA5DVS/6 may be contacted at 1196 Phillips Court, Santa Clara, California 95051. James may also be reached by e-,mail at jwbennett@sprintmail.com.

AmQRP.ORG HOMEBREWER

Ken Newman, N2CQ

QRP Contesting Calendar

January 2005

40 METER FOXHUNT - Every Friday 0200z to 0359z
(Thurs 9 PM to 11 PM EST)
Info: http://www.cqc.org/fox/index.htm

Truffle Hunt - 30 min before Fox Hunt
Info: http://fpqrp.com/winter_hunt.html

AGB NYSB -"New Year SnowBall" Contest (80M - All) ...QRP Category
Jan 1, 0000z to 0100z
Rules: http://www.qsl.net/eu1eu/agb_nysb.htm

AGCW Happy New Year Contest (CW)...QRP Category
Jan 1, 0900z to 1200z
Rules: http://www.agcw.de/english/contest/happynew_e.htm

Original QRP Contest (CW - 80, 40 & 20m) ... QRP Category
Jan 1, 1500z to Jan 2, 1500z
Rules: http://www.qrpcc.de/contestrules/oqrpr.html

Adventure Radio Society - Spartan Sprint (CW) ... QRP Contest!
Jan 4, 0200z to 0400z (First Monday 9 PM EST)
Rules: http://www.arsqrp.com/

The World QRP Federation (WQF) QRP Party (All) ... QRP Contest!
Jan 7, 0000z to 2400z
Rules: http://ruqrp.narod.ru/wqf_e.htm

North American QSO Party (CW) (100w max. QRP Entries Noted)
Jan 8, 1800z to Jan 9, 0600z
Rules: http://www.ncjweb.com/naqprules.php

ARRL RTTY Roundup ... Low Power Category
Jan 8, 1800z to Jan 9, 2400z
Rules: http://www.arrl.org/contests/calendar.html?year=2005

070 Club PSKFEST Contest ... QRP Category
Jan 15, 0000z to 2400z
Rules: http://www.podxs.com/html/pskfest.html

Michigan QRP Club Contest (CW) ... QRP Contest!
Jan 15, 1200z to Jan 16, 2359z
Rules: http://www.qsl.net/miqrpclub/contest.html

LZ OPEN CONTEST (CW 80M/40M) ...QRP Category
Jan 15, 1200z to 2000z
Rules: http://www.qsl.net/lz1fw/lzopen/index.html

North American QSO Party (SSB) (100w max. QRP Entries Noted)
Jan 15, 1800z to Jan 16, 0600z
Rules: http://www.ncjweb.com/naqprules.php

Run For The Bacon (CW) *** QRP Contest ***
Jan 17, 0100z to 0300z
Rules: http://fpqrp.com

ARRL January VHF Sweepstakes ... Low Power Category
Jan 22, 1900z to Jan 24, 0400z
Rules: http://www.arrl.org/contests/calendar.html?year=2005

CQ WW 160-Meter DX Contest (CW) ... QRP Category
Jan 29, 0000z to Jan 30, 2359z
Rules: http://www.cq-amateur-radio.com/awards.html

UBA DX Contest (Belgian) (SSB) ... QRP Category
Jan 29, 1300z to Jan 30, 1300z
Rules: http://www.sk3bg.se/contest/ubac.htm

Thanks to K3WWP, LA9HW, SM3CER, WA7BNM, ARRL and others for assistance in compiling this calendar.

Anyone may use this "QRP Contest Calendar" for your website, newsletter, e-mail list or other media as desired. (Of course, please include a credit to the source of this material.)

QRP Contesting "Tips & Techniques"

I may not have anything new in contesting but it wouldn't hurt to review on these points. Anyone with more to add, PLEASE DO, and we'll add your input to the review.

ANTENNAS, ANTENNAS and ANTENNAS. Many would agree that antennas are the first three thing to improve our results in contesting for any kind, whether Sprint or Contest.

Sample of the above:

QRP Afield '96 was won easily by AC5AM, Bob Stolzle. Having been in the contest, we wondered how it could be that this guy was S9 the whole contest, and always there sending 950 MW power. Now we know. The complete story was published in the "72" of Jan. '97. Again, antennas. He found the highest point in the area which just happened to have an old, unused broadcast radio tower. He raised two full wave delta loops (20 and 40 meters) and inverted vee's (20/40 mtrs) at the 100' level. A coax switch is used to choose the best antenna for the incoming signals.

The 100' level probably made the difference, being a strong signal from a QRPp station. His point too is that resonant antennas do much better than those with tuners. The idea was to call CQ on the loop. If no answer, call CQ on the Vee, and so on. If answered on a weak signal, switch to the other one if received better.

Now on to the rest of the real QRP World. Not many are going to find a unused 100+ foot tower to raise these types of antennas. Given that for the Contest type of event, the next thing would be the Op.

For the Op, the biggest thing would be the amount of operation time he is having fun with. Very few put in the 24 hours for the QRP ARCI Spring and Fall contests. The ones in the "TOP TEN" list reached that level according to the hours spent, assuming they are more than average stations. How do they do that? First one have to have the desire. It becomes less desirable at the 15 hour level unless you had plenty of sleep before the contest. To stay there, you have to have the necessary operator comforts. You know what they are for you. Things like a comfortable chair and headphones will take you further.

The first thing that took me to longer operating duration was the memory keyer. With this you can be relaxed much longer and your fist won't give out.

In recent years, computer logging will go even further. I use "NA" for the QRCI contests. When calling CQ, you can set a repeat every few seconds. All you need do is listen. If a station is answering, press the "ESC" key to stop the repeats, and enter his callsign. Press the "INSERT" key, and the station is answered and your info is sent. (You can input an RST if you wish). Enter his info and press the "=" key and you have sent an OK and a QRZ. That is it. (Except the log that is printed at the end.) Given that, you could be there a much longer time.

What else for the operator? **Practice, practice, and practice**! The more you do the better you get.

There are some excellent links to more contest tips. I would recommend the K3WWP link first. If you have been in contesting, chances are you worked John. He is using a homebrew QRP tube transmitter with an indoor antenna. If you don't have the antenna like AC5AM had, you can relate to K3WWP. Look for his tips at:
http://home.alltel.net/johnshan/contest_ss_tips.html

After that, you may want to check the tips from the top contesters. Check them out at: http://www.contesting.com This is not QRP but many of the suggestions will help us all in contesting. This covers the pre-contesting, equipment, operating, and post-contesting. Most tips assume that you are running a KW, beams and contest software.

Anyone have more QRP contesting tips? I'm sure all would like to hear about them and we'll add any that can help us along.

CU in the next 'test!

72/73,
Ken Newman, N2CQ
Woodbury, NJ
n2cq@arrl.net

Doug Hendricks, KI6DS

Farewell to a Friend

QRP has lost an Icon. Jim Cates, WA6GER, cofounder of NorCal QRP Club, died from a bout with Cancer this past April. Jim was a huge contributor to QRP, and the best friend that I have ever had. He was an advisor, confidant, mentor, Elmer, sage and most of all, always my friend.

We started NorCal in 1993 in order to find 15 or 20 guys who would be interested in sharing QRP. There was never an intent to grow into a world wide club, or “go global” as Jim told Jay Bromley one time. It just happened. Jim Cates was the reason NorCal was like it was. He was behind the scenes, but he had a tremendous input and influence on things the club did. He came up with most of the features of the NorCal 40, he insisted that every kit be first class, and most of all, he said that we should serve QRPers.

Neither Jim nor I ever made a penny from NorCal. In fact it cost us money. All of the trips that Jim made to promote QRP were made at his expense. He was the money man behind NorCal. Jim handled the funds, he paid the bills, I spent the money, a great partnership.

Pacificon was Jim’s idea. He wanted to have a forum like Dayton on the west coast, so that guys out here would not have to make the expensive trip to Dayton in order to enjoy a QRP forum. He came up with the idea in 1993, but we did not get to implement it until after FDIM was held the first time. But we were planning Pacificon before FDIM was announced. I remember Jim’s comment when FDIM was announce on QRP-l. He said that “Great minds think alike.”

Jim was inducted into the QRP Hall of Fame in 1997, and there has never been a more deserving member. Jim’s actions changed QRP. He mailed every single NorCal kit over the years, except for the 49er, which was shipped during his wife’s terminal illness. Jim was caring for his wife, and did not have time to do QRP. He felt bad about it, but his duty called him, and he was there. When I was nominated for the hall of fame, I was going to refuse it because Jim was not elected with me. I told Jim that in a packet, and pretty quickly the phone rang. Jim told me that I was not going to refuse, because it would dishonor him. I didn’t understand, but did when Jim explained that he had taken his name out of consideration. He insisted that I accept, for NorCal, if not for my self. Reluctantly I agreed. Jim told me later that it was one of his proudest moments when I was inducted into the hall of fame. When he was inducted, I was privileged to introduce him. I asked him afterwards what he felt, and he said, “Embarrassed”.

When Jim's wife Electra was dying from cancer, Jim took care of her at home, night and day, for months. He told me at the time that if he ever got cancer, that he would not want to have visitors, and that he would want to die with dignity. He said that would include my visiting him, and he made me promise to not come see him if he ever got cancer. I thought it was a strange thing to say, but I agreed. Jim sent an email to Paul Maciel and me in January of this year. I will never forget it. It said, "There is no easy way to say this so here it is. I have cancer and it is moving very fast. There is nothing that can be done. I am sending you the club money, and will send Paul the extra kits that I have. Thanks for all that you have done for me. Jim."

I was in shock. And I was in a quandary. My best friend in the world was dying, and I had promised to not go see him. I would never see Jim again. It was hard, but I never discussed it with anyone. The word did get out to a few friends, and some were probably upset that I did not tell them about Jim, but I had given my word to respect his privacy and I was keeping it. A few QRPers dropped by to see Jim and visited him, and then told me about it. All of them had a strange look when I told them that I had not been to see Jim. But they didn't know about my promise. Then in March, I got an email that I will never forget. Steve Cates, Jim's son, said that Jim had changed his mind and wanted to see me. I immediately called, even though it was late on a Sunday night. Steve assured me that it would be best to come up on Saturday. I contacted Paul Maciel, and asked him to go with me. He readily agreed.

My last visit with Jim Cates was sad, yet it gave me peace. Jim was in a hospital bed, and was too week to sit up or even roll over by himself. But he was in full control of his mind. He was concerned about QRP, and wanted to make sure that his illness was not causing any problems for Paul or me. He apologized for asking us to come see him, but said that he was wrong when he asked me not to come see him, and that he appreciated our coming so much. That was Jim Cates, always thinking of others, even on his death bed. Jim also kept his sense of humor to the end. He told Paul and I that there was a conflict of interest at his house. We asked what that was, and Jim replied. "I am trying to die, and my sons are trying to keep me from dying. But I am going to win!!". Paul and I both chuckled and we knew that he still had his sense of humor.

Jim Cates and I were partners for 11 years. During that time we sold thousands of kits, saw NorCal grow into a world wide organization, and become a household word among QRPers. We never had a cross word with each other, never an argument or even a disagreement on anything. I had Jim's credit card, and was authorized to spend up to 25,000 dollars on it. Now that is trust. We trusted each other. Jim made a rule when we started NorCal. It was simple. Each of us would have veto power on anything the club did. If one of us wanted to do something and the other didn't, it would not happen. Neither of us ever used that veto power, not even once. His wife, Electra, used to say that we were the odd couple. Jim was Tony Randall, polite and mannerly, I was Walter Mathau, go for broke, impetuous, never met a stranger. We were opposites, and we were a team.

Jim was honored by me once before, when he appeared on the cover of QRPp. Jim was the only person ever to have their picture on the cover of QRPp, and that was by design. I had to get his wife to help with getting the picture, and Electra thoroughly enjoyed being in on the fun.

I miss Jim terribly. The years that I knew him he was a father to me. I am much better for knowing him. QRP without him will go on, but it won't be the same without that quiet, unassuming gentle man sitting in the background, just enjoying being there. As our friend Jon Iza says, "Be well my friend, be well."

Doug Hendricks, KI6DS may be reached by mail at 862 Frank Avenue, Dos Palos, CA 93620, or by email at ki6ds@dpol.net

New! KX1 Ultra-Portable Transceiver

- Weighs 9 oz.
- 2 or 3 bands
- Up to 4 W
- DDS VFO
- SWL Receive
- Internal ATU
- Internal battery

Our KX1 CW transceiver kit is the new featherweight champ! With all controls on top, it's ideal for trail-side, beach chair, sleeping bag, or picnic table operation. And at 1.3"H x 5.3"W x 3"D, it's truly pocket size. Its superhet receiver covers ham *and* nearby SWL bands; the variable-bandwidth crystal filter handles CW, SSB, and AM. Also features memory keyer, RIT, 3-digit display, audible CW frequency readout, and a white LED logbook lamp. The internal battery provides *20 to 30 hours* of casual operation. Add our KXPD1 paddle and KXAT1 automatic ant. tuner options to create a fully-integrated station. KX1 basic kit covers 20 & 40 m ($279). KXB30 adds 30 m ($29).

K1 and K2 Transceivers

The compact K1 4-band, 5 watt+ CW transceiver kit is great for first-time builders, draws only 55 mA on receive, and makes a great travel radio. The K2 transceiver kit offers incredible receiver performance, all-band SSB/CW coverage, & optional DSP. It can be configured for 100 watts when the going gets rough. Both transceivers have internal battery and automatic antenna tuner options. Please visit our web site for details on our full line of high-performance kits.

www.elecraft.com

ELECRAFT

P.O. Box 69
Aptos, CA 95001-0069

Phone: (831) 662-8345
sales@elecraft.com

VISA MasterCard

Super Antennas!

email: w6mma@superantennas.com

http://www.superantennas.com

1606 Pheasant Way - Placerville, Ca 95669

1-530-622-6668

Portable HF Antennas

(7-450 Mhz) QRP to 150 Watts

3.5-4mhz with optional coil (only with MP-1)

Drive-on Antenna Mount

New!

5.5-21' Collapsable Mast

Portable Yagi System

Portable Dipole System

New!

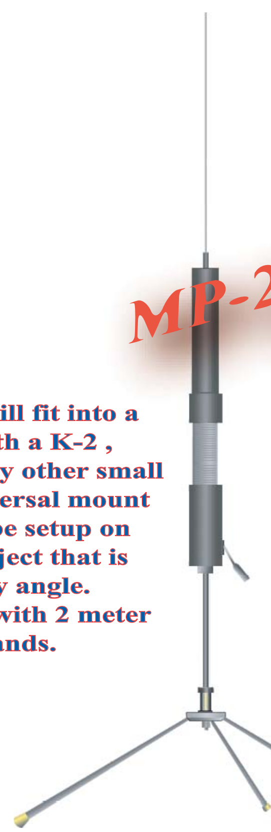
SDC 100

Screw Drive Controller

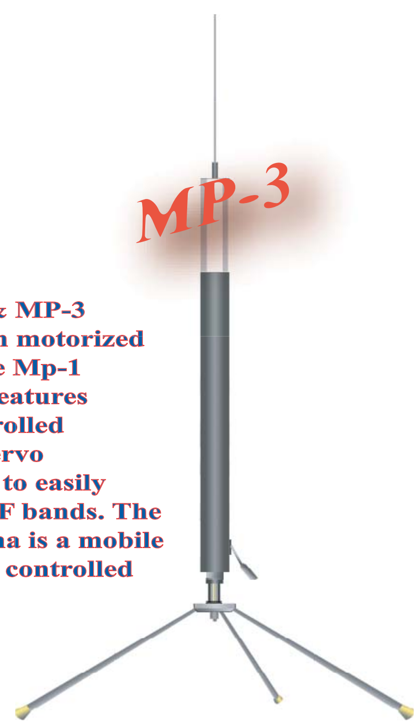
MP-1

The MP-1 will fit into a briefcase with a K-2 , FT817 or any other small rig. It's universal mount allows it to be setup on most any object that is handy at any angle. Also works with 2 meter and 70cm bands.

MP-2



MP-3



The MP-2 & MP-3 antenna is an motorized hybrid of the Mp-1 antenna. It features remote controlled motorized servo mechanisms to easily adjust the HF bands. The MP-3 antenna is a mobile mount servo controlled