Construction Manual
for the …

K8IQY 'Precision *VXO*'

Designed by: **Jim Kortge, K8IQY**

Kitted by: **The NJQRP Club**

K8IQY Precision VXO

The Precision VXO kit is designed to provide a precise and stable frequency source for crystal parameter analysis, and as a tool for accurately sorting crystals for use as filter elements. It can also be used in any application where a stable frequency source is needed, and for which a crystal is available on or near the required frequency. Suggested other uses include a receiver alignment source, DC receiver local oscillator, or the first stage of a QRP transmitter.

Introduction

The parts supplied in the kit contain everything needed to assemble the unit except a power connector. This part was not supplied, since everyone seems to have his own favorite. A Ten Tec TP-17 case is included. However, the case is not drilled nor labeled; the builder must perform those operations. Templates are provided to assist with those construction steps.

Assembly is broken up into logical steps, so that each section can be built and tested. While that approach is recommended, it is not necessary, as the PC board is logically laid out and well marked with the component locations. No alignment operations are required after assembly.

A parts list is included in this construction manual. Upon opening the kit, one should take inventory and verify that all of the parts were supplied. If any are missing, please contact the New Jersey QRP Club for replacements (David Porter, AA3UR, aa3ur@comcast.net). Please also check the project's web page for late breaking news, construction tips, application notes, etc. (www.njqrp.org/pvxo). Technical questions concerning the design may be directed to the designer (Jim Kortge, K8IQY, jokortge@prodigy.net).

Theory of Operation

The schematic for this Precision VXO design is shown in figure 1.

This generator employs a Colpitts oscillator, Q1, using a 2N5484 junction FET. The crystal is usually one from a set that will be used for building a filter or for Local Oscillator service. However, if the Precision VXO is being used for receiver alignment, the crystal might be one whose frequency is inside one of the lower ham bands. Controlling the frequency of oscillation is accomplished with a varicap diode, a MV209, and a set of 5 molded inductors configured as a binary weighted set. Inductance needed to force the crystal to oscillate at its marked case frequency is experimentally determined by selecting the desired total inductance using switches S1 through S5. When no inductors are selected, the minimum inductance is that of the leads going to S1 and S5. Under those conditions, the crystal will oscillate at its highest frequency. As more inductance is added, the crystal's oscillating frequency moves lower. Maximum inductance is obtained when all of the inductors are selected and are connected in series. Under these conditions, many higher frequency crystals may cease to oscillate. The minimum inductance change is 2.2 uH, provided by S1.

With most crystals, the tuning range of the varicap diode can provide the overlap needed to make the digitally controlled inductance work seamlessly. It can also provide sufficient frequency span so that the resonant frequency of another crystal under test can be determined. Frequency precision is obtained by using a 10-turn potentiometer, and stability is achieved through the use of an internal regulated supply, and by minimizing the RF levels in the oscillator circuit.

A second 2N5484, Q2, is used as a source follower to further isolate the oscillator from any changes reflected back from the output load. The stage is also set up, by way of a voltage divider (R8 and R9), to provide an output impedance of 50 ohms. This impedance properly drives the fixed input impedance of the Norton amplifier used in the output stage. Both Q1 and Q2 are supplied with 9 volts from a regulator. Using a regulated supply helps maintain frequency and amplitude stability.

The output stage, a PN2222 (Q3) provides a gain of +12 dB and amplifies the generated signal to a level of approximately –10 dBm, or 250 millivolts peak-to-peak. This stage is a noiseless Norton amplifier design. Downstream of this amplifier is a low pass filter, to reduce harmonic content above 20 MHz. A spectrum plot of the generator's output is shown in figure 2.

Overall, this Precision VXO generator is designed to work with commonly available HC49, and HC49U style computer crystals, often used in IF filter service. It will operate with crystals in the frequency range of 3.5 through 13.5 MHz, but is optimized for crystals at 9 MHz and below.

Besides having a clean output spectrum, this low cost generator also has excellent frequency stability. Its warm-up drift is a few Hz, and the stability over a 24-hour period is within 20 Hz. Short-term stability is not measurable with the equipment in the K8IQY laboratory.

The typical tuning range with a 4.9152 MHz crystal is plus and minus 250 Hz, more than adequate for covering the series resonant frequency of crystals being characterized, or matched. More importantly, with the wide range of inductance available, the oscillating frequency of the VXO

crystal can be moved over a considerably wider range, assuring that one can find the series resonant frequency of a crystal under test.

PC Board Preparation

Before starting the assembly of the PC board, a few important steps should be undertaken. The PC board supplied may have excess substrate material on the front edge, extending beyond the copper layer on the bottom. This extra material should be removed, allowing a better fit of the PC board into the case. The excess material can be removed with a file, sander, or with a shear if one is available. Using a "tin snip" type cutter is not recommended, as it may over flex the board, and permanently distort it. Be careful with the board so as not to damage it.

Once the excess material is removed, the board is ready to be used as the template for marking the case for drilling. Drilling will be done later in the construction process. If the case marking is not done with the unpopulated board, it can't be done accurately later.

Case Marking

Begin this phase by removing the case from its wrapper. Separate the upper half from the lower. Cover the upper half top surface with white paper, cut to size, and taped to this case half. Place the bare PC board on the paper, and center it in all four directions, so that the gaps on the left and right edges match, and the gaps to the front and rear match. Tape the PC board in place also. Using a sharp pencil, mark the four outer holes of each switch position, S1 through S5. Next, mark the three holes of the crystal socket. Carefully un-tape the PC board, but leave the white paper taped to the top case half. The markings applied will be used later for accurately drilling this case half.

In a similar manner, tape a piece of white paper to the front of the lower case half. With the front edge of the PC board aligned with the bottom of the case, and the board aligned side to side, tape the PC board in place. Using a pencil again, mark the 3 terminal holes for the 10-turn potentiometer, Pot1, and the two larger terminal holes for the BNC connector, J1. Carefully, un-tape the PC board, but leave the paper taped to the front face of the lower case half. The marked locations will also be used later for drilling this case half.

PC Board Assembly

Our assembly approach will be to follow the schematic diagram of the Precision VXO, and essentially build from it in terms of which parts are installed, and the order they are installed. While this method isn't as detailed as the old "Heath kit process" of calling out a part, installing it, and then checking off a box, it is more than sufficient for this simple kit. The schematic is included as figure 1.

Please note that all of the components mount on the top or component side of the PC board except the 10-turn potentiometer (Pot1) and the BNC connector (J1). Those two components mount on the bottom, or solder side of the board. If you decide not to follow the suggested assembly, leave the installation of the 10-turn potentiometer and the BNC connector until all of the "component side" components have been completed. This makes handling the PC board easier without those two large components being on the underside.

Start by stuffing the board with the **Power Supply Protection** components. These include the solid-state fuse (SSF1) and the 1N4744A Zener diode (D4). Make sure the cathode end of D4, marked with a bar, matches the silk-screened symbol on the PC board. Solder these components and clip off their excess lead length.

Next, install the **+9-volt Regulator** components. These components are 0.1 uF capacitors C1, C3, and C5, 10 uF capacitor C2, and 100 uF capacitor C4. Also, 100 ohm resistor R2, and regulator U1. Place the body of the regulator up off the surface of the PC board about ¼ of an inch, and orient it to the outline on the PC board. Solder these components and clip off their excess lead length.

If you want to test the assembly at this point, apply 12-13.8 volts DC to the "Pwr" and "Gnd" pads on the PC board. Make sure you have the polarity of the applied voltage correct. Using a DVM, verify the voltage at the junction of R2 and C4 is 9 volts.

Continue on by installing all of the components that comprise the **VXO** part of the circuit in the following order. Group 1: 100 pF capacitors C6 and C7, 4.7 pF capacitor C8, 30 pF capacitor C9, and 0.1 uF capacitor C10. Solder these components and clip off their excess lead length. Group 2: 100 uH inductor L1, 2.2 uH inductor L2, 4.7 uH inductor L3, 10 uH inductor L4, 22 uH inductor L5, and 39 uH inductor L6. Solder these components and clip off their excess lead length. Group 3: 100 K resistor R3, 47 K resistor R4, and 1.8 K resistor R5. Solder these components and clip off their excess lead length. Group 4: 2N5484 transistor Q1, 1N4148 diode D1, and MV209 varicap diode D2. Make sure that these parts are oriented to match their silk-screened symbols and are elevated ¼ inch above the PC board surface. Solder these components and clip off their excess lead length. Install the 3-pin crystal socket Y1. The leads for this socket should just extend from the bottom of the PC board, and should be perpendicular on the topside. Solder the center pin only so the junction can be reheated if the orientation needs adjustment. When it is properly aligned, solder the remaining outside pins. Install DPDT switches S1-S5. Make sure each switch is in contact with the surface of the PC board and perpendicular to the top surface before soldering its pins. Install one switch at a time and work slowly. Removing a switch or attempting to reorient it after several of its pins are soldered will probably destroy the PC board.

If you want to test the assembly at this point, apply 12-13.8 volts DC to the "Pwr" and "Gnd" pads on the PC board. Make sure you have the polarity of the applied voltage correct. Using a DVM, verify the voltage at the junction of R2 and C5 is 8.6 volts. If you have an RF probe or oscilloscope, plug a 3.5 to 9 MHz crystal into socket Y1. Place the handle of all 5 switches toward the rear of the PC board. Place the RF probe or oscilloscope probe on the junction of L1, C6, and C7. There should be several volts p-p (3 volts p-p for reference) of RF signal present. Remove the crystal, and verify the RF voltage drops to near zero. If may

not drop to zero as the circuit may continue to oscillate at 300+ MHz at low amplitude.

Continue the assembly by installing the components for the **Source Follower**. Group1: 10 pF capacitor C11, and 0.1 uF capacitors C12 and C13. Solder these components and clip off their excess lead length. Group 2: 100 ohm resistor R7, 100 K resistor R6, 560 ohm resistor R8, and 56 ohm resistor R9. Solder these components and clip off their excess lead length. Group 3: 100uH inductor L7 and 2N5484 transistor Q2. Make sure Q2 is oriented correctly, and elevated above the PC board surface about ¼ inch. Solder these components and clip off their excess lead length.

If you want to test the assembly at this point, apply 12-13.8 volts DC to the "Pwr" and "Gnd" pads on the PC board. Make sure you have the polarity of the applied voltage correct. Using a DVM, verify the voltage at the junction of R7 and C12 is also 8.6 volts. If you have an RF probe or oscilloscope, plug a 3.5 to 9 MHz crystal into socket Y1. Place the handle of all 5 switches toward the rear of the PC board. Place the RF probe or oscilloscope probe on the junction of R8 and R9. There should be approximately 100 millivolts p-p (116 millivolts p-p for reference) of RF signal present. Remove the crystal, and verify the RF voltage drops to near zero.

Before building the last section of the generator, transformer T1 for the output amplifier should be constructed. This transformer is wound with a total of 15 turns, tapped at the 4th turn from the beginning of the winding. Start this winding using about 12 inches of #26 or #28 wire. Bring an end through the toroid core from the backside, and bend it over the top edge of the core in the 1 o'clock position, with about 1½ inches of the free end beyond the edge of the core. Next, grab the free end that is sticking back through the core, and bring it forward, push it through the core hole, and pull it tight. The turn just wound should be below, or clockwise around the core from the free end at the 1 o'clock position. This is the 2nd turn, as one turn is counted each time the wire passes through the center of the core. Wind two more turns in the same manner as before. The wire that has the "working end" should now be behind the back edge of the core. Pull it off to the right at the 3 o'clock position, and make a loop that is about 1½ inch out and another 1½ inch back, with the two wires parallel to each other. Twist the loop several times to hold these two wires until the twists come up to the back, outside edge of the core. This loop will be the tap at the 4th turn. Continue winding in the same direction until another 11 turns has been placed on the core. The exiting wire will be at the 7 o'clock position, on the backside of the core. When you are done, cut the remaining wire off leaving a 1½-inch end. Save the cut off wire, as it will be used for the one turn link when the transformer is installed. Scrape or burn off the insulation from the 3 ends, and tin with solder. A page is included in this manual containing pictures of the process just described.

Form the leads of the core so that the 4-turn winding and 11-turn winding will line up with their respective pads on the PC board. Place the core on the PC board with the three leads through the correct pads, and bend the leads outward at a 45-degree angle to keep the core positioned.

Take the remaining scrap of wire used for winding and remove the insulation from one end for about ½ inch. Skip the next 1 inch, and remove the insulation from the next one inch, and cut the wire there, creating a piece 2½ inches in length. Tin the bare areas, and solder the ½ inch end into the pad that also connects to the emitter of Q3 and inductor L8. Route this wire up over the left edge of the toroid core, down through center of the core, and under the right edge, with the remaining end going through the hole in the pad which is also connected to capacitor C14. The core will probably have to be elevated some to get the end of the wire through the pad hole. Once this is done, pull the 3 leads going to the 4 and 11 turn windings taut, and solder them. Finally, pull the remaining end of the one turn link taut, and solder it.

The remaining parts for the **12 dB Amplifier** can now be installed. Group 1: 0.01 uF capacitors C14 and C16, 0.1 uF capacitors C15 and C17, and 220 pF capacitors C18 and C19. Solder these components and clip off their excess lead length. Group 2: 100 ohm resistor R10, 5.6 K resistor R12, and 47 ohm resistor R11. Solder these components and clip off their excess lead length. Group 3: 100 uH inductor L8, and 0.68 uH inductor L9. Solder these components and clip off their excess lead length. Group 4: PN2222A transistor Q3, and green LED D3. D3 is installed with the longer lead, the anode, going through the pad that is also connected to the junction of Q3's base, C15, and R12. Place Q3 and D3 above the PC board surface ¼ inch. Solder these components and clip off their excess lead length.

If you want to test the assembly at this point, apply 12-13.8 volts DC to the "Pwr" and "Gnd" pads on the PC board. Make sure you have the polarity of the applied voltage correct. Using a DVM, verify the voltage at the junction of R11 and C17 is 13.1 volts with 13.8 volts applied to the PWR pad. If you have an RF probe or oscilloscope, plug a 3.5 to 9 MHz crystal into socket Y1. Place the handle of all 5 switches toward the rear of the PC board. Place the RF probe or oscilloscope probe on the junction of L9 and C19. There should be about 500 millivolts p-p (510 millivolts p-p for reference) of RF signal present. Remove the crystal, and verify the RF voltage drops to near zero.

The output voltage will drop when the generator is driving a 50-ohm load. Its output under that load should be about 250 millivolts p-p, or 90 millivolts RMS, or –8 dBm, or 160 microwatts, should you decide to use it as a transmitter.

The last two components can now be installed. Both the 5 K – 10-turn potentiometer, Pot1, and the BNC output connector, J1, are installed on the solder, or bottom side of the PC board. The threaded parts of these components are used to hold the PC board to the front of the bottom case. The switch set is used to hold the case top to the bottom.

Before mounting the potentiometer, remove the "ears" from each of the terminals by cutting parallel to each edge. When this operation is completed, each terminal will be the same width from its top end, to the body of the potentiometer, and

will fit through the hole in the solder pad. Put the potentiometer in place on the bottom of the board with the shaft pointing outward, and make sure it will seat down onto the lower surface. Once in place, solder the center terminal from the top, or component side of the board. Make sure the pot shaft is perpendicular to the front surface edge of the PC board, and then solder the remaining two connections.

Mount the BNC connector by inserting it through the 4 mating holes from the bottom, or solder side of the PC board. Make sure that is seats tight against the surface, and solder one connection on the top, or component side of the PC board. After assuring it is perpendicular to the front surface edge of the PC board, solder the remaining 3 connections.

Congratulations, assembly of the PC board is complete! If you have done the interim testing along the way and that was successful, your Precision *VXO* is also fully functional.

Case Drilling

Drilling the two case halves should be done with care, as once a hole is drilled, it cannot be easily moved. Starting with the upper case half, performing the following steps. Using a sharp pencil and a straight edge, mark lines between the outside, diagonal pairs of holes that were places on the white paper covering. Each of these elongated Xs becomes the drill center for one of the inductance switches, S1 through S5. If you have done the layout correctly, there should be ½ inch between adjacent switches. If your measurements do not match this dimension, determine where the error is and correct it. The hole centers must be ½ inch apart for the top cover to fit properly over the five switches. Drill these holes at either ¼ inch for a tight fit, or 9/32 for a more tolerant fit.

The 0.1 X 0.3 inch rectangular hole for crystal socket Y1 is made by drilling each of the 3 points marked on the white paper with a number 37 or 38 drill. Before drilling, verify the spacing between these 3 marks is 0.100 inch and the marks are in a straight line. Drill the outer two holes first, and then the center location. Finish the opening by cutting the web away between the 3 holes with a utility knife, and finish the opening with a small flat file.

If you have done the drilling accurately, the case top will fit over the switches, and the crystal socket will protrude from the rectangular cutout. The nuts and washers need to be removed from the switches for a proper fit. A switch lock washer may be needed on each switch as a shim during final assembly.

Now take the bottom case half and perform the following operations on it. Using a sharp pencil, draw a line through the center of the 3 locations taken from Pot1. You may want to use a combination square for this task, using the top edge of the case as the horizontal reference. Once this line is drawn, mark a cross line exactly 1 inch below the top case edge. This locates where the 3/8-inch diameter hole will be drilled for accepting the bushing of Pot1. Carefully drill that hole.

Using the two marks taken from the BNC connector, find the center between these two locations, and mark that point. Using a combination square set on the top edge of the lower case half, mark a vertical line through the center point just determined. Measuring down from the top edge 7/8 inch, place a cross mark on the vertical line. This locates where the ½ inch diameter hole for the BNC connector will be drilled for accepting the bushing of that connector. Carefully drill that hole also.

Finally, decide what type of power connector you prefer, and drill an appropriately sized hole on the backside of the lower case half. A suggested location is 1 inch down, and 7/8 inch over from the right edge, as you are viewing that panel surface. That location will place your power connector opposite the location of Pot1, on the front panel.

After all of the holes are drilled, remove the white paper drilling templates from each case half. Then deburr each hole using a deburring tool, or a utility knife. Trial fit the PC board into the lower case half, securing it with the mounting hardware supplied with the 10-turn potentiometer, and the BNC connector. Only tighten the nuts finger tight. Place the top case half on, and place mounting hardware on the outer two switches. Tighten these nuts finger tight also. If the crystal socket is protruding more than 1/16 of an inch vertically in its opening, remove the top cover and add a lock washer to each switch shaft, and reassemble. The crystal socket exposure should be correct now.

At this point, the case halves can be cleaned, primed and painted. Or you may use your favorite finishing method. Included in this manual are templates for suggested lettering of the controls, but you may prefer something more elegant.

Printed-paper templates on photo quality paper using an ink jet printer provided the lettering on the prototype. Each lettering template was cut to size and affixed face down to Avery brand self-adhesive laminating sheet, part number LS10P. The paper backing was replaced for handling, and each assembly cut to size. With the backing removed, the templates were affixed to the front and top panels of the lower and upper case halves respectively. A hobby knife was used to remove the covering where the controls and switches mount.

Final assembly

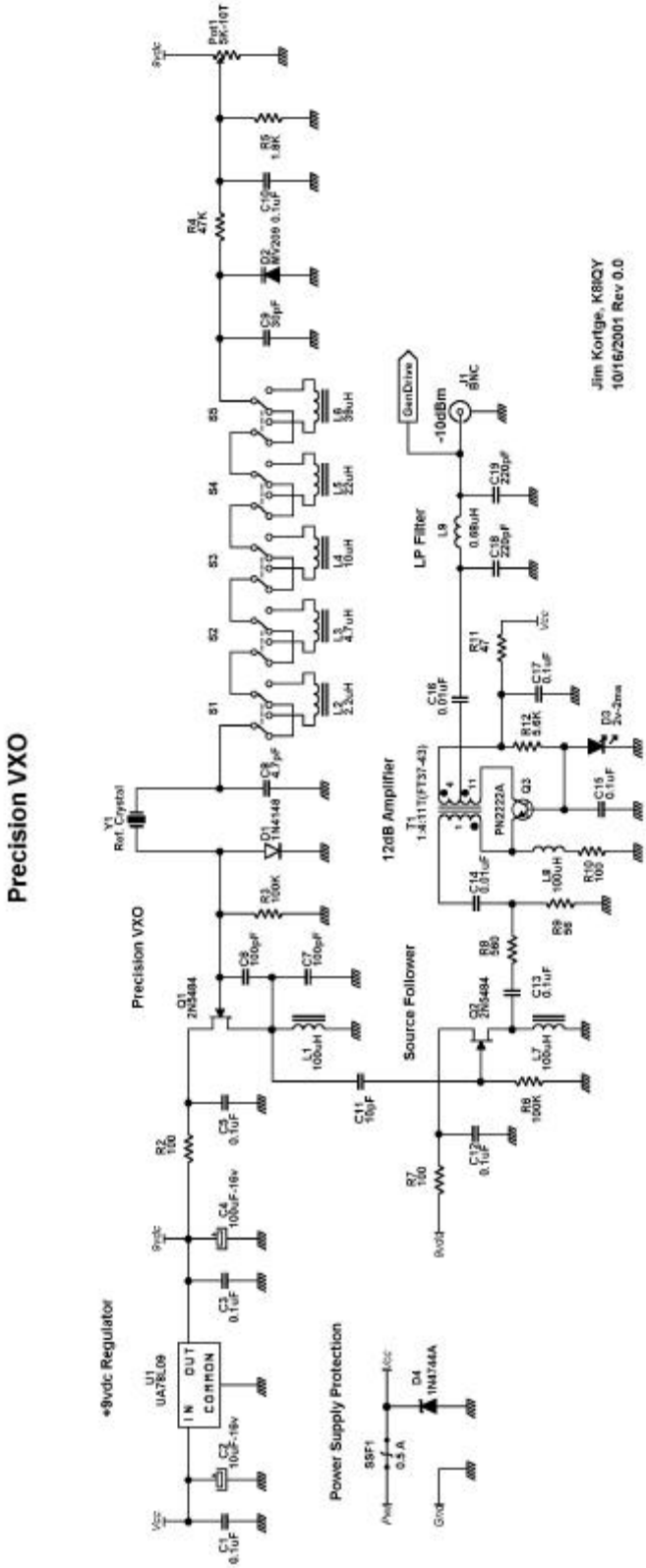
Final assembly entails soldering a pair of wires between the power pads on the PC board and your power connector. On the prototype, the power connector mounts from the inside, so it was wired to the PC board with approximately 4-inch leads. The PC board is then installed in the lower case and the potentiometer and BNC connector control nuts gently tightened. Next, the power connector is installed with its mounting hardware. Finally, the top case half is brought down over the switch shafts, making sure the crystal socket is protruding, and the mounting nuts tightened. None of the mounting nuts need to be much more than hand tight. Too much torque could distort the PC board if the fit-up in your case isn't good.

That's it…your Precision VXO is finished!


Frequency
Output
Precision VXO
NJ QRP Club


Xtal
Inductance
Out
In
39uH
22uH
10uH
4.7uH
2.2uH
Frequency
Precision VXO

Precision *VXO* Schematic Diagram – Figure 1



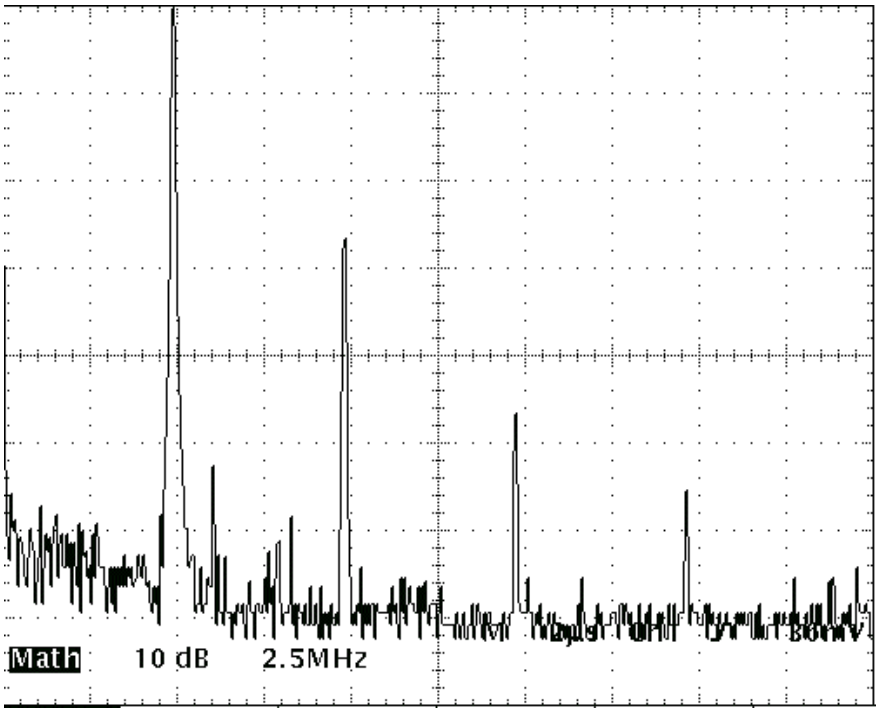
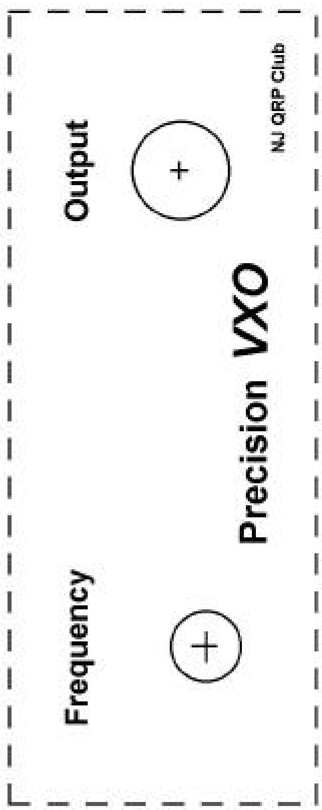
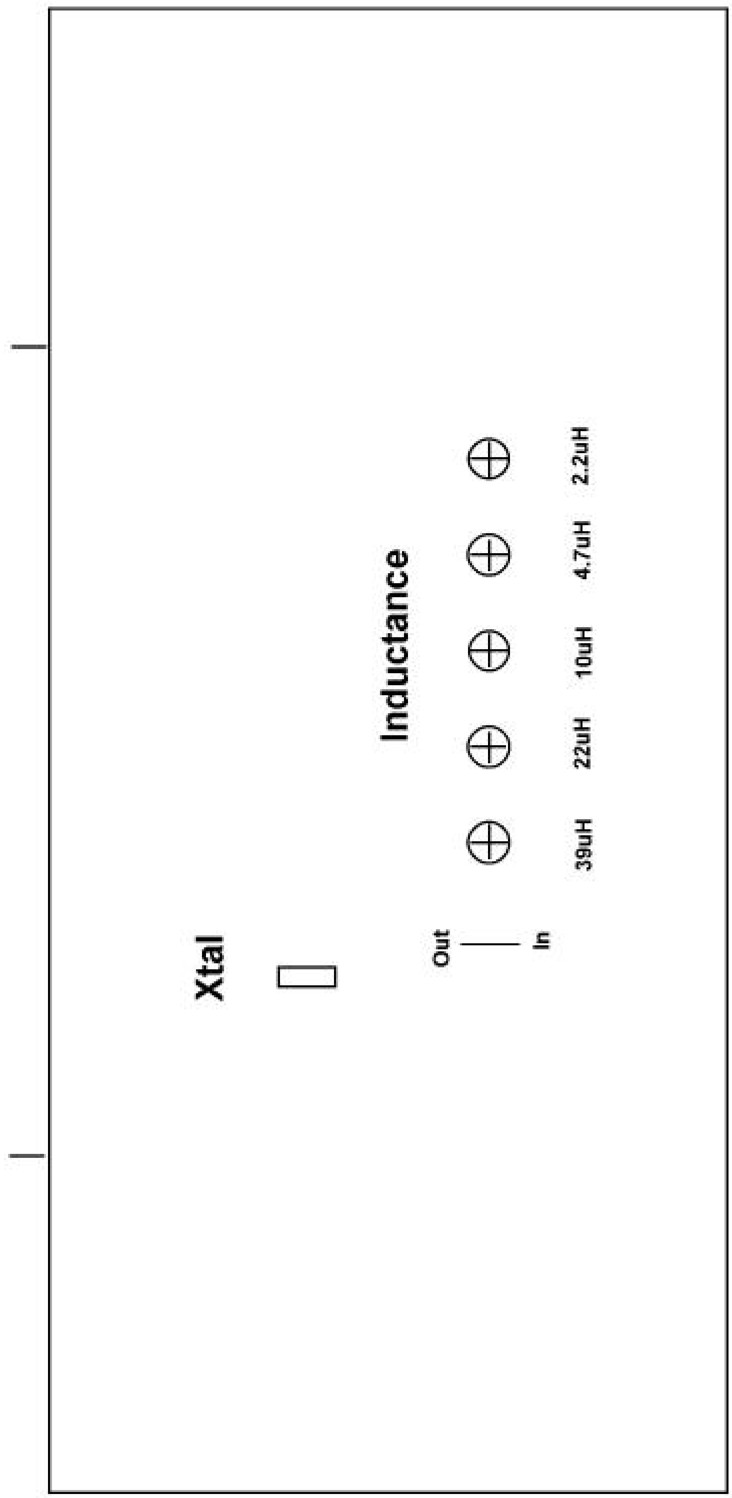


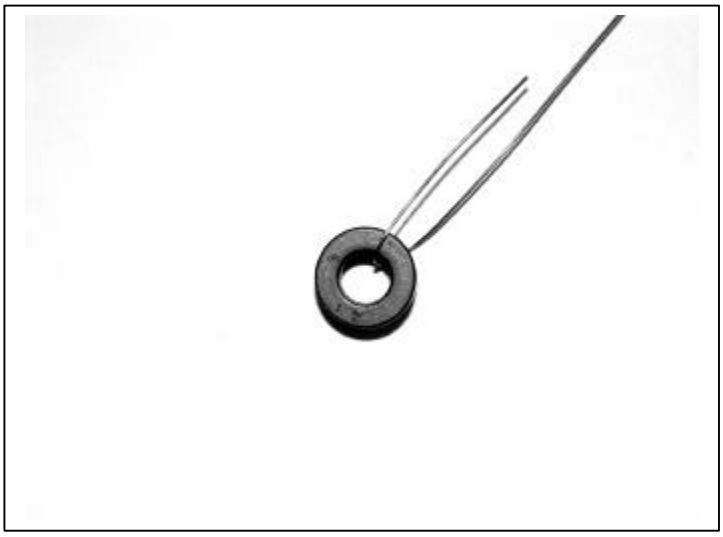
Figure 2

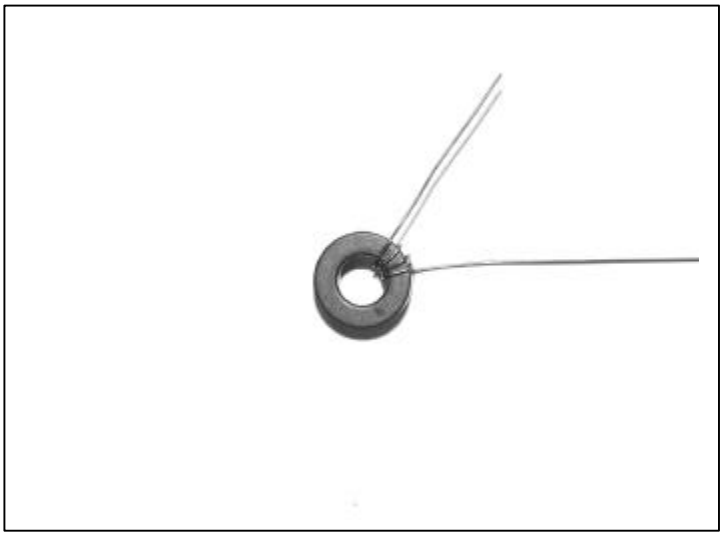


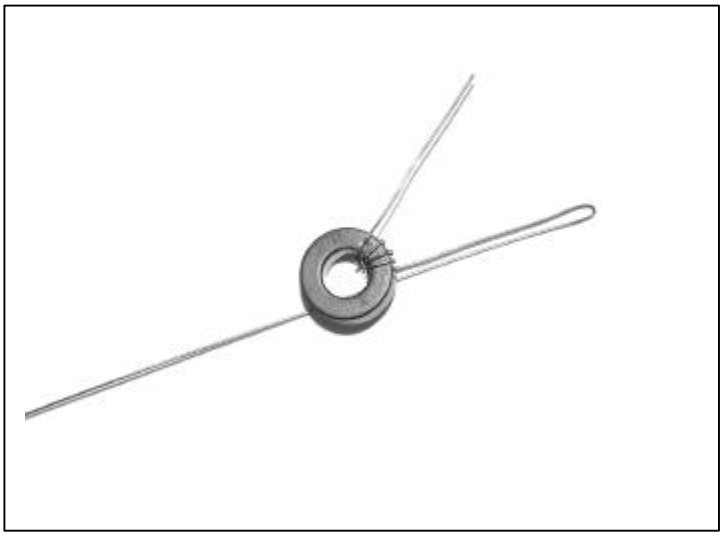
Front Panel Lettering Template

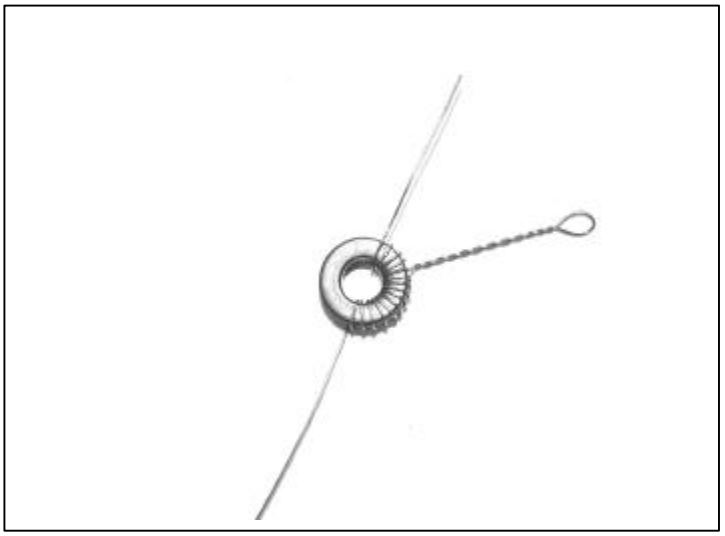


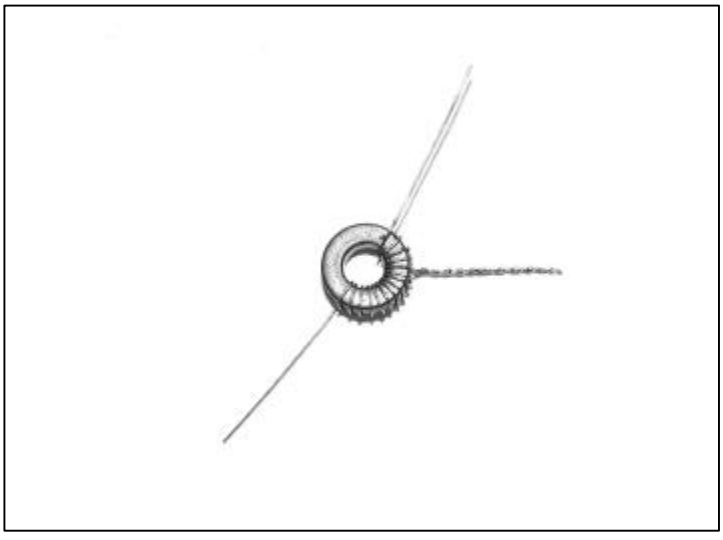
Top Panel Lettering Template

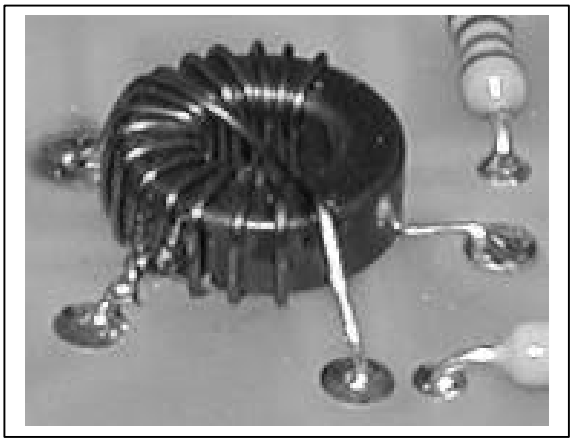






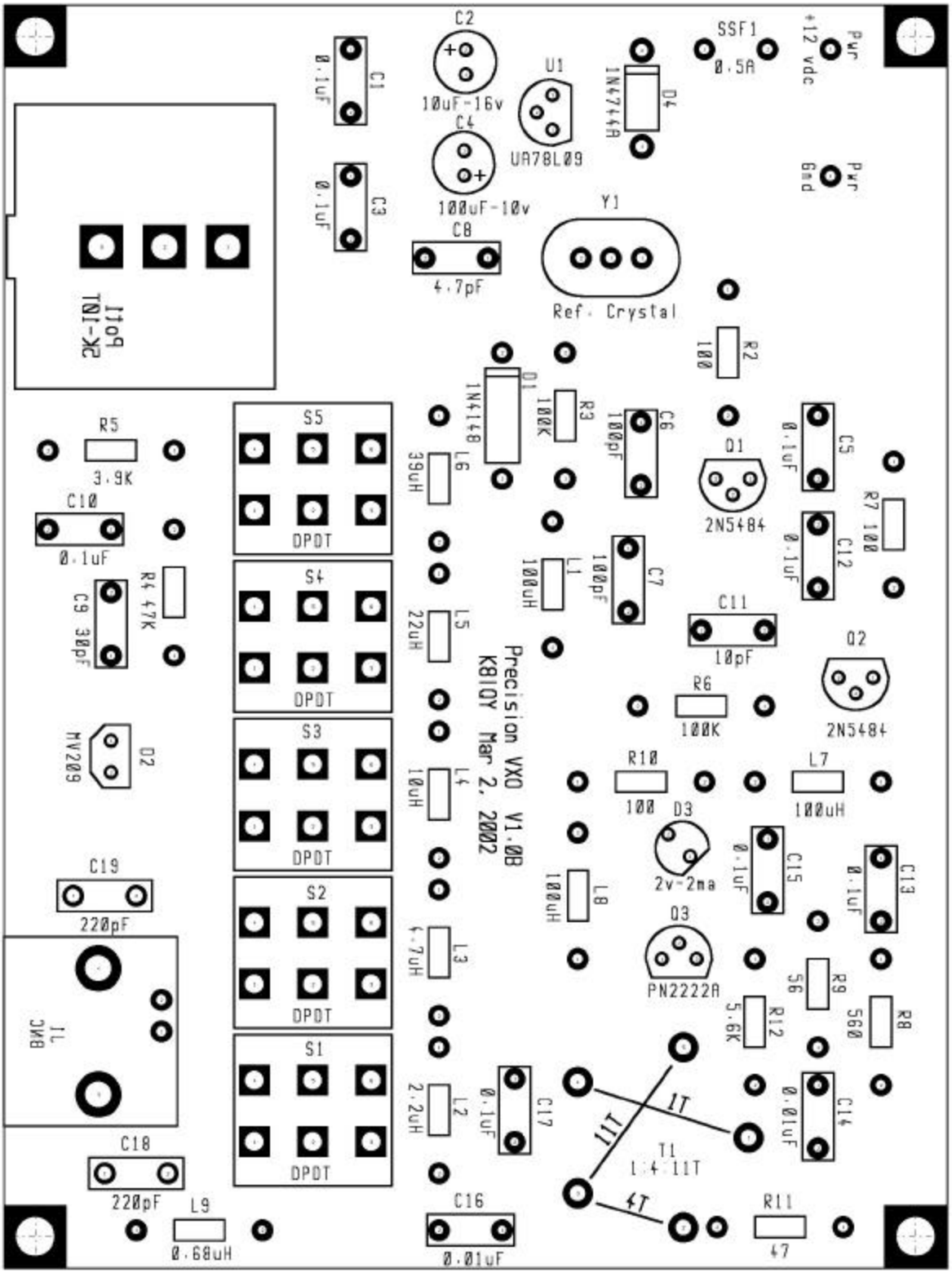






Transformer T1 winding example

PC Board Layout



Bill of Material

Item	Qty	References	Value	Description
Capacitors				
01	1	C8	4.7pF	NPO cap (4.7)
02	1	C11	10pF	NPO cap (10)
03	1	C9	30pF	NPO cap (30)
04	2	C6,C7	100pF	NPO cap (101)
05	2	C18,C19	220pF	Disc cap (221)
06	2	C14,C16	0.01uF	Disc cap (103)
07	8	C1,C3,C5,C10,C12,C13,C15,C17	0.1uF	Mono cap (104)
08	1	C2	10uF-16v	Electrolytic cap
09	1	C4	100uF-16v	Electrolytic cap
Resistors				
10	1	R11	47 ohm	Res (Yel Vio Blk)
11	1	R9	56 ohm	Res (Grn Blu Blk)
12	3	R2,R7,R10	100 ohm	Res (Brn Blk Brn)
13	1	R8	560 ohm	Res (Brn Blu Brn)
14	1	R5	1.8K	Res (Brn Gry Red)
15	1	R12	5.6K	Res (Grn Blu Red)
16	1	R4	47K	Res (Yel Vio Org)
17	2	R3,R6	100K	Res (Brn Blk Yel)
18	1	Pot1	5K-10T	10 turn pot (Blu)
Inductors				
19	1	L9	0.68uH	Ind (Blu Gry Sil)
20	1	L2	2.2uH	Ind (Red Red Gld)
21	1	L3	4.7uH	Ind (Yel Vio Gld)
22	1	L4	10uH	Ind (Brn Blk Blk)
23	1	L5	22uH	Ind (Red Red Blk)
24	1	L6	39uH	Ind (Org Wht Blk)
25	3	L1,L7,L8	100uH	Ind (Brn Blk Brn)
26	1	T1	1:4:11 T	FT37-43 toroid
Diodes/transistors/regulator				
27	1	D1	1N4148	Signal diode
28	1	D2	MV209	Varicap diode
29	1	D3	2v-2ma	LED (Grn)
30	1	D4	1N4744A	Zener diode
31	2	Q1,Q2	2N5484	FET transistor
32	1	Q3	PN2222A	Bipolar transistor
33	1	U1	UA78L09	Voltage regulator
Miscellaneous				
34	1	SSF1	0.5 A	Fuse (Yel, marked B050)
35	1	J1	BNC	Connector (Blk)
36	5	S1,S2,S3,S4,S5	DPDT	Switch (Blu)
37	1	Y1		Crystal 3 pin socket
38	1	--	------	Precision *VXO* PC board

NOTES

Kit designer:

Jim Kortge, K8IQY
PO Box 108
Fenton, MI 48430-0108

email: jokortge@prodigy.net

Kitted & Produced by:

The New Jersey QRP Club
George Heron, N2APB
2419 Feather Mae Ct
Forest Hill, MD 21050

email: n2apb@amsat.org

Technical Support:
Jim Kortge, K8IQY email: jokortge@prodigy.net

Missing Parts, Assembly Questions, etc:
Dave Porter, AA3UR email: aa3ur@comcast.net

PVXO News, App Notes, etc:
NJQRP Website: www.njqrp.org/pvxo