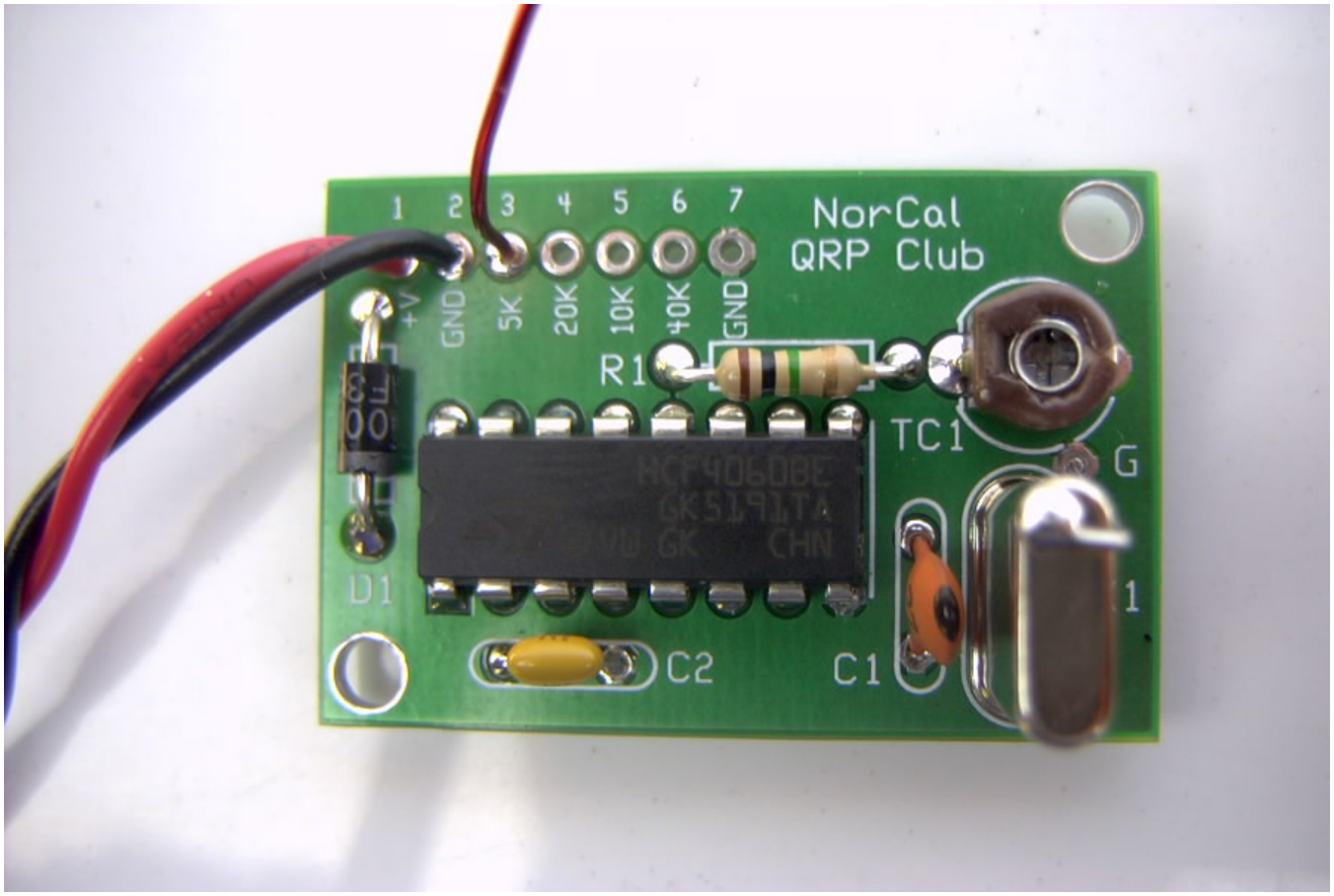
The VE3DNL Marker Generator

Designed by Glen Leinweber, VE3DNL



Kitted by the NorCal QRP Club

Manual Copyright 2006, NorCal QRP Club

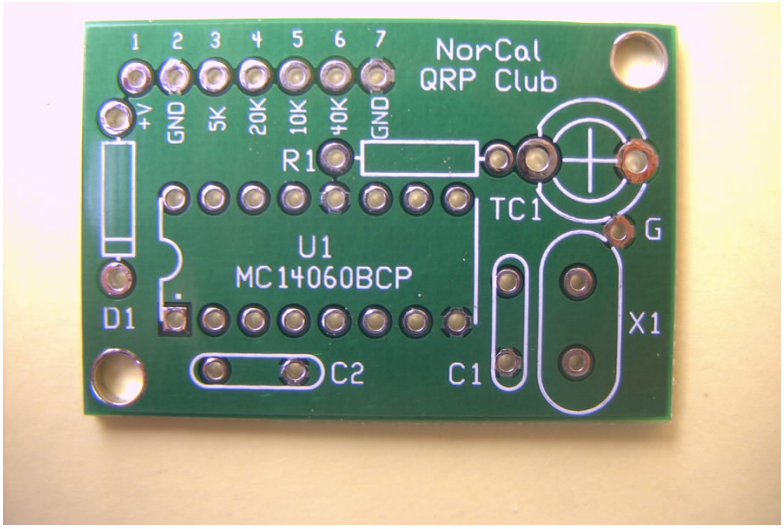
Written by Doug Hendricks, KI6DS, Edited by Paul Maciel, AK1P

NorCal QRP Club VE3DNL Marker Generator

In July of 1995, Glen Leinweber, VE3DNL, posted to QRP-L a simple circuit diagram for a crystal controlled marker generator. Chuck Adams, K7QO, took the diagram and laid out a printed circuit board and Far Circuits did a small pc board for it. In 2001, the Fort Smith QRP Group, which was selling the kit at that time, decided to upgrade the board to include a 1N4001 diode to provide reverse polarity protection. Dave Fifield, AD6A, provided the new layout, and NorCal assisted the Ft. Smith QRP Group with the project. In 2002 FSQRP Group decided to discontinue the kit, and sent the remaining boards back to NorCal. NorCal sold the approximately 100 kits, and then discontinued the kit until now. We are bringing back the VE3DNL Marker Generator and hope that you enjoy building and using your kit.

The circuit consists of a single CMOS 16 pin integrated circuit manufactured by Motorola, the MC14060. This IC is an oscillator with a binary divider to divide the oscillator frequency down by a power of 2. For example, if we use a crystal with a series resonant frequency of 5.120 MHz and divide it by 512 we get a frequency of 10 kHz, division by 1024 yields 5 kHz, etc. There are a number of outputs for these frequencies. They are 5, 10, 20 and 40 kHz. Since these outputs are nearly square waves there are a large number of harmonics present up to approximately 30 MHz and these are used as "markers" that we can hear in a receiver.

You will find a printed circuit board that is 2.5 x 3.75 cm, with white printing on the side of the board on which the parts are placed. Orient the board with the text "NorCal QRP Club" in the upper right hand corner as you look at the board from the top, i.e. the lettering side. All the instructions below will be done with the board oriented this way.



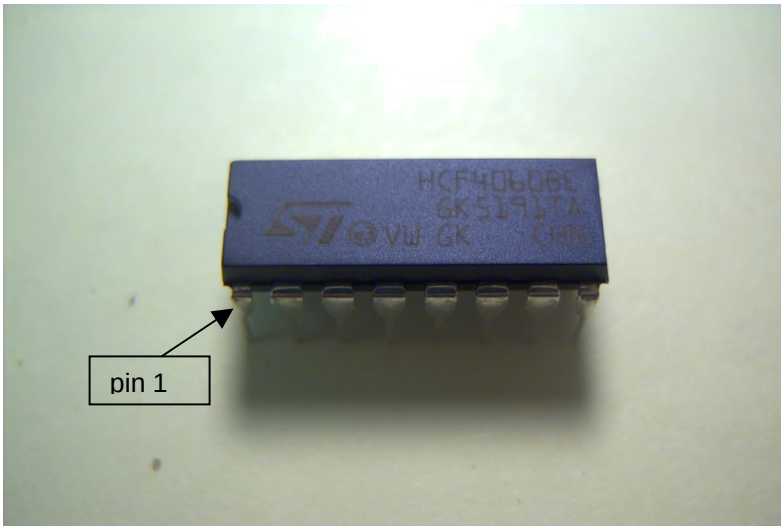
Bare NorCal VE3DNL Board showing proper orientation for the instructions.

You should read chapter 25 of a recent edition of the ARRL Handbook if this is your first building experience. This is the chapter on circuit construction and has details on how to solder and the tools you will need. You will need the following tools to build this kit: soldering iron (30 W Maximum), solder used for electronics, needle nose pliers, diagonal cutting pliers, and a wire stripper if you use insulated wire for the output.

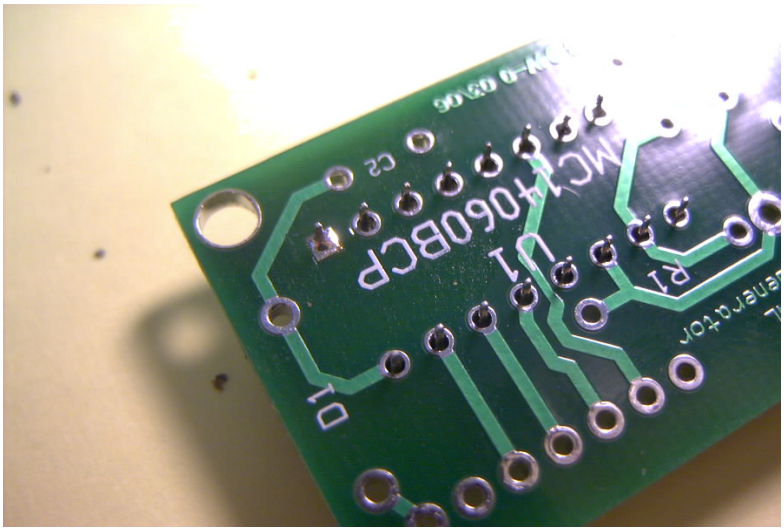
The IC chip is a CMOS device so anti-static techniques should be used to prevent damaging the part. This means that you avoid generating static electricity that will destroy the chip internally. Chapter 25 of the ARRL Handbook has the details. You shouldn't have to go to such extreme measures if you are careful not to generate electrical charges as you work on the circuit. The PC board is marked to show where pin 1 of the chip goes. There is a square on the pin outline of the chip. When you install the chip, make sure that the dot on pin 1 of the chip and the half-moon cutout on the chip are aligned with this square. Another way to think of it is that pin 1 goes toward the left side of the board towards the diode.

Now let's start building.

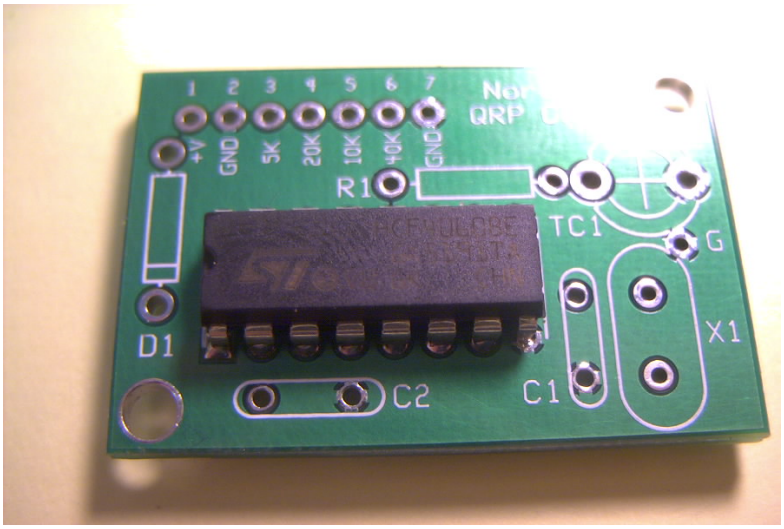
Step 1: Locate the IC Chip that is in the parts bag. It is pictured below. Note that if you look carefully you can see the notch that is on the left side of the chip.



Install the IC chip first with pin 1 close to the 0.1uF capacitor, C2. Use care in installing the chip. You may use a socket if you wish, but one is not needed or provided with the kit. Just take your time and insert the IC into the board. Make sure it is almost flat against the board and solder one of the corner pins. Check to make sure that pin 1 is oriented correctly, and solder the remaining pins.

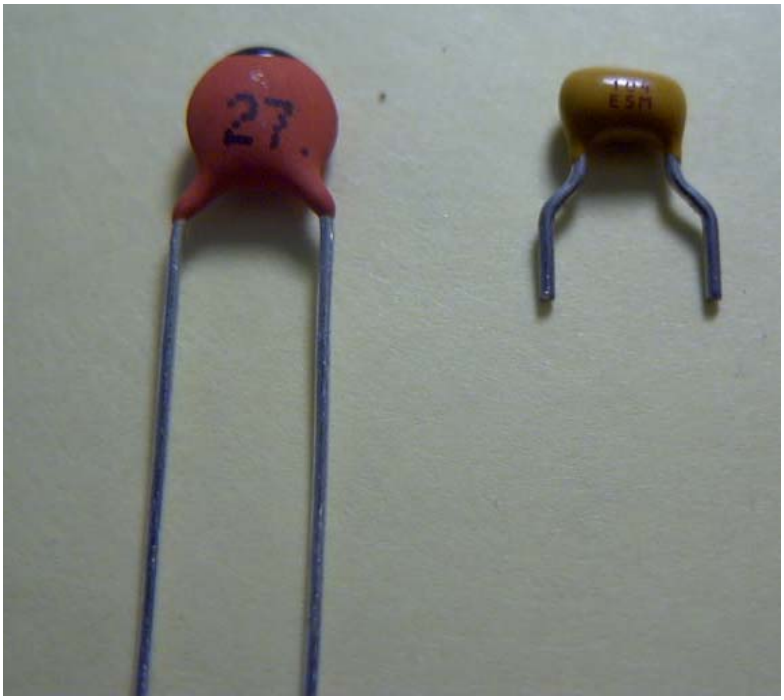


Underside of VE3DNL showing the first pin of the chip soldered. Not that the outline of pin 1 is square, not round like the rest of the pins.

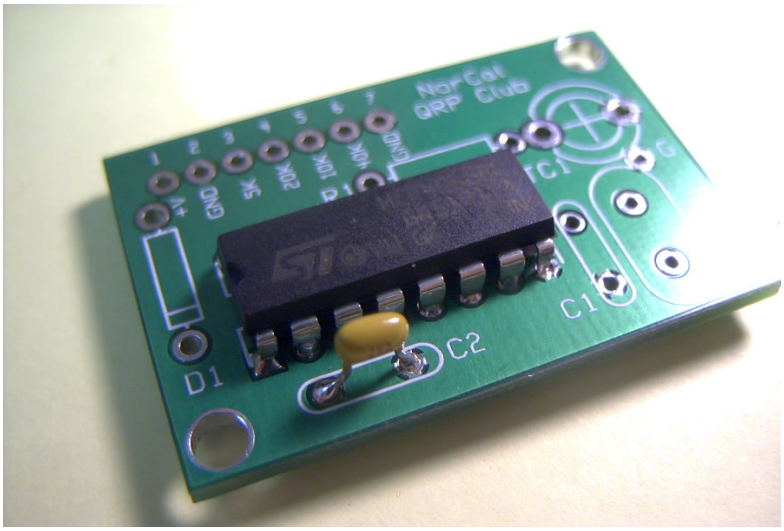


IC1 installed in the VE3DNL board. Note that the notch should be to the left in this view.

Step 2: Next install the 0.1uF capacitor, marked with 104 at C2.

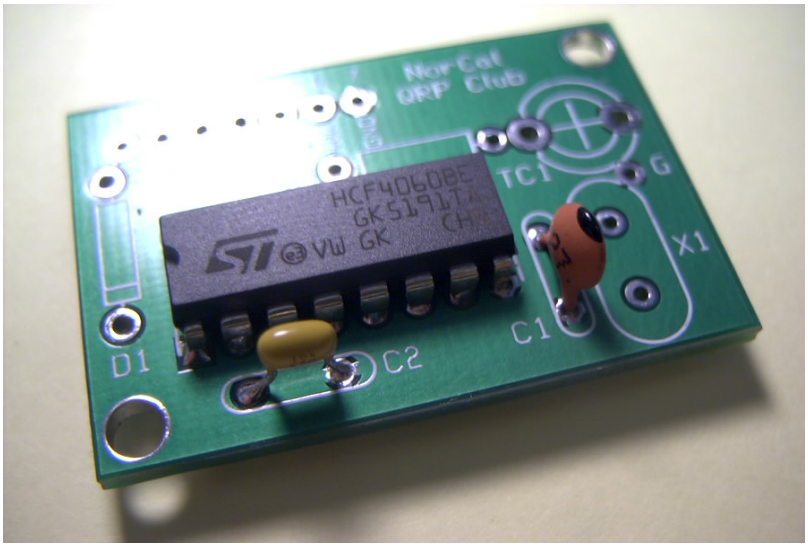


There are 2 capacitors in your kit. The one on the left with the long leads and black dot is a 27pFand is C1. The one on the right is a 0.1uF, and is C2.



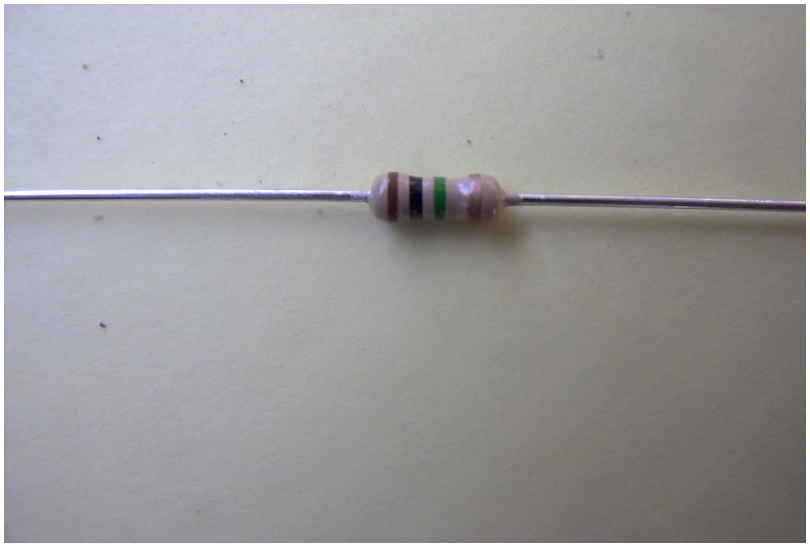
C2 has been installed.

Step 3: Find the 27pF disc capacitor marked 27 with a black dot on top. Solder it in just to the right of the IC at C1.

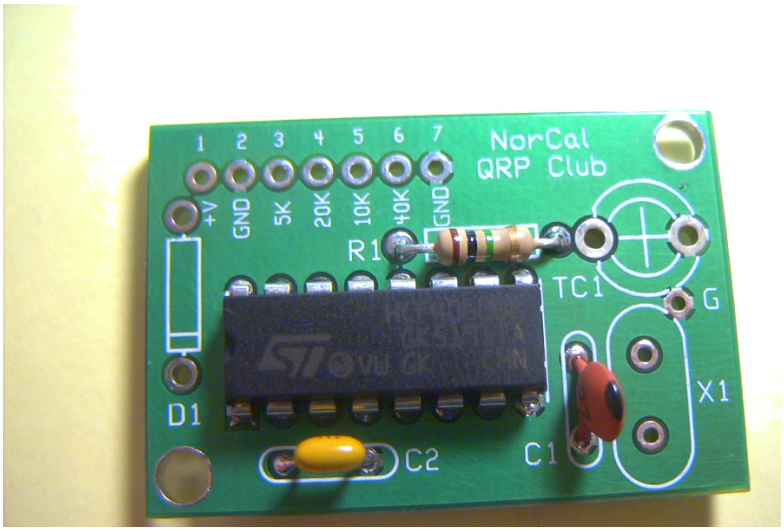


C1 has been installed.

Step 4: Now find the 1 Meg resistor color coded brown-black-green-gold, and solder it in where the 1 Meg resistor outline is on the board, R1. You will note that there are colored bands on the resistor. This identifies the value of the resistor. There is no right or left to a resistor, so you can orient it either way, but most people like to orient them so that they read from left to right as viewed from the bottom of the board. In this case, that means that we have the brown-black-green-gold bands mounted as shown. Brown means 1, Black 0, Green X6 and gold tolerance of 5%. Thus this is a 1000000 ohm or 1 meg resistor with a tolerance of 5%



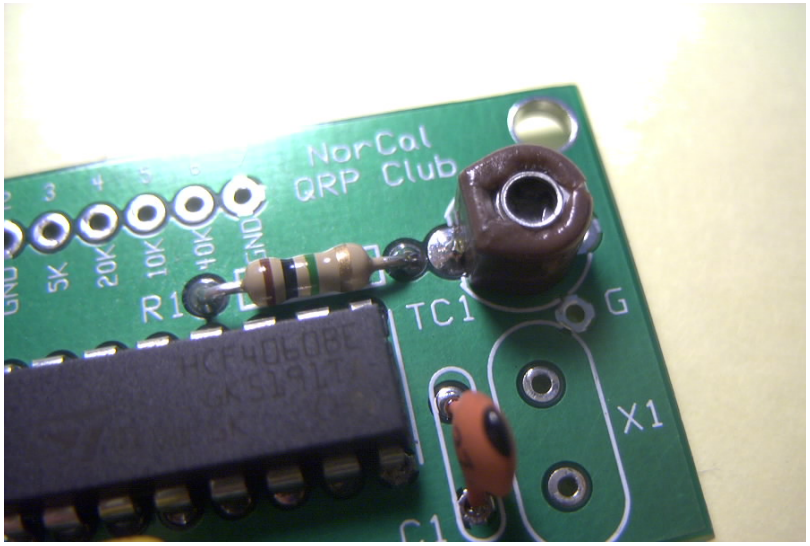
Resistor R1



The resistor R1 is now installed.

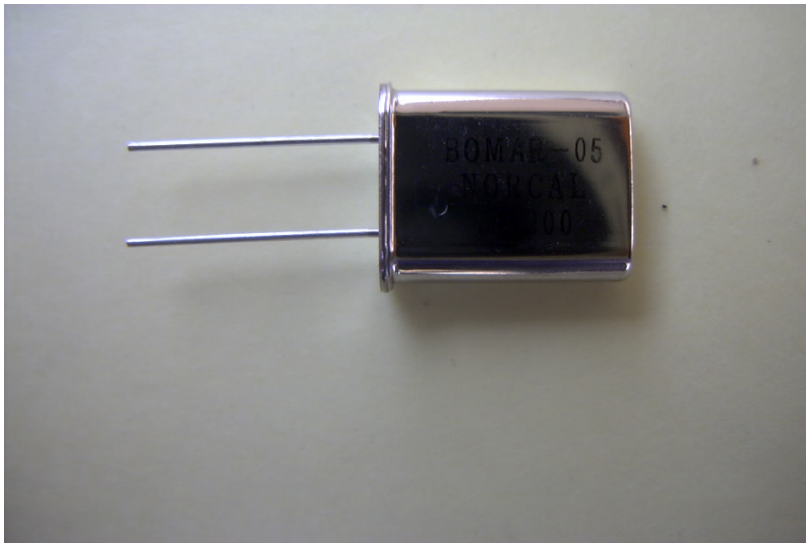
Step 5: The trimmer capacitor will be installed next. It is a brown circular cap with two leads, and has a slot on the top to adjust it with. There is a flat side of the resistor as shown. You will solder it in the circled area labeled TC1. It is important that you orient the trimmer with the flat side towards the IC.

Trimcap, TC1. Note that there is definitely a flat side of the cap.

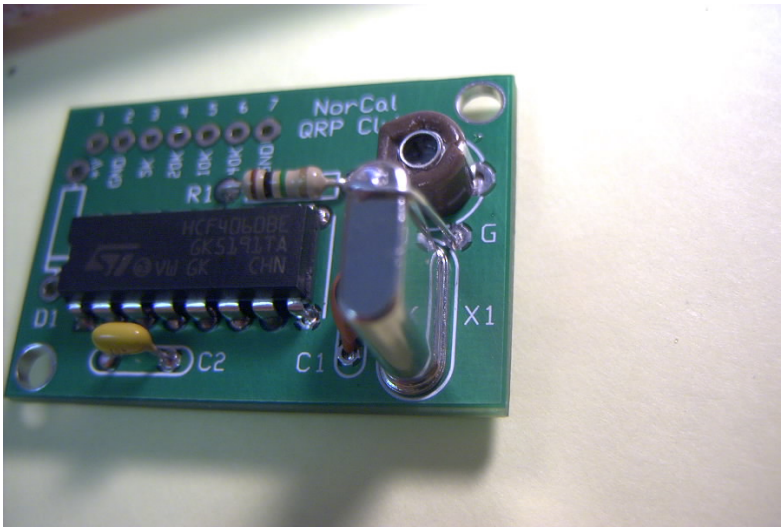


TC1 installed.

Step 6: Install the 5.120 MHz crystal next to the 27pF capacitor, C1 at X1. It should sit flat on the board. Use a cutoff resistor lead to ground the crystal case using the hole marked G. Place the cutoff resistor lead in the hole, solder, and then bend the lead over the top of the crystal and solder it to the crystal. Do not use too much heat.

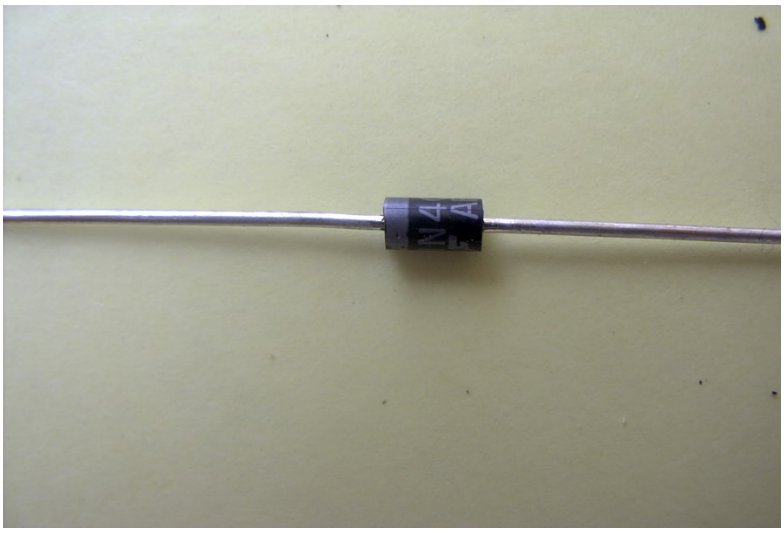


NorCal 5.120 MHz Crystal



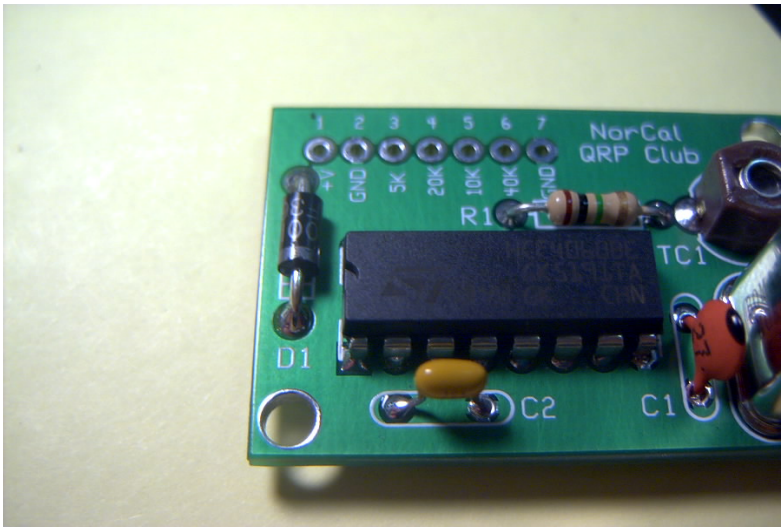
X1 installed with ground lead from the top of the crystal to hole G. Be sure to solder the lead to the top of the crystal and to hole G.

Step 7: Next we will install a diode. Diodes have polarity; it is indicated by a band on the diode. Note in the picture below the band on the left side of the diode. This is the banded end.



Diode D1. Note polarity band on left end.

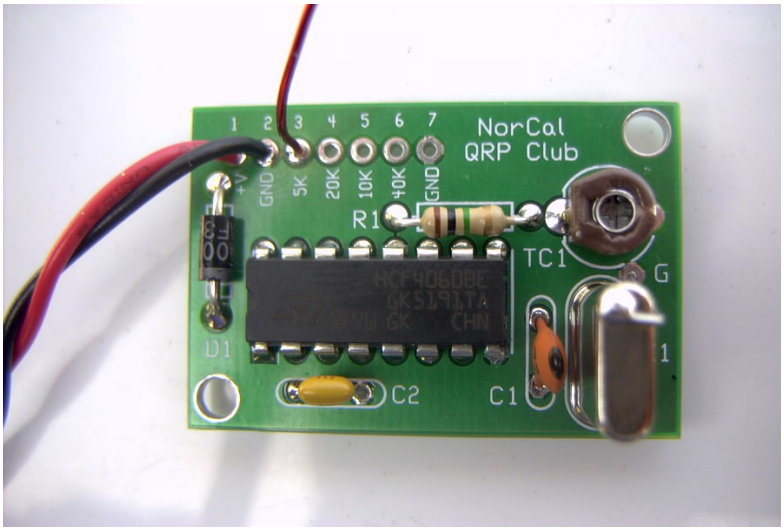
Place Diode D1 as shown, marking sure to put the banded end as indicated on the parts layout.



Diode D1 installed. Note that the banded end is towards the bottom of the board. This is important.

Step 8: Now attach the 9V battery connector at the points labeled +V (1) and GND (2) on the board. The Red lead of the connector goes to the +V, the black lead of the connector goes to GND. You may need to remove about ¼" of insulation from each lead of the connector and tin them before soldering them in.

9V Battery Lead



9V battery leads and wire antenna installed.

Step 9: You will need to provide some type of "antenna" for your generator. You may use machined IC pins as a socket for each of the outputs and use a single wire to put in the socket. Or, you may wish to use a switch to choose the output. This is up to you. The simplest method is to put a wire about 6" long in the 5 kHz output and solder it. But this restricts the use to only one output.

Step 10: That's it. You now have a working signal generator and frequency marker. The simplest way to test it to see if it works is to connect a 9V battery, and then hold your generator near a receiver tuned to say 3.900 MHz. If you don't have a ham band receiver, you can tune an AM receiver to 1600 kHz. Simply hold the marker generator near the radio. You should hear a tone in the receiver. If you don't, try turning the variable capacitor. Note when mounting your kit that you should use plastic or nylon spacers or if you use metal, be sure to use a non-conducting washer between the board and spacer so that you will not short a trace.

Step 11: Calibration. You have several choices on how to calibrate your kit. If you own a frequency counter, measure the frequency at pin 9 of the IC and adjust for 5.120 MHz using the trimmer cap. If you do not own a frequency counter but have a shortwave receiver capable of receiving WWV at 5, 10, or 15 MHz, then you can "zero beat" the signal from WWV and the signal generator to calibrate the frequency output. There are a number of uses for the marker generator, and here is a partial list:

Signal Generator
Crystal Calibrator
Signal Source for Receiver Alignment
Signal Source for Dial Calibration
Reference Source to locate frequency on a receiver

Parts List

C1 – 27pF disc marked 27
C2 - .1uF mono cap, marked 104
TC1 – 5 – 50 pF trimmer cap (brown)
D1 – 1N4001 diode
Circuit Board
9V Battery Connector
R1 – 1Meg (brown-black-green-gold)
U1 – MC14060 IC