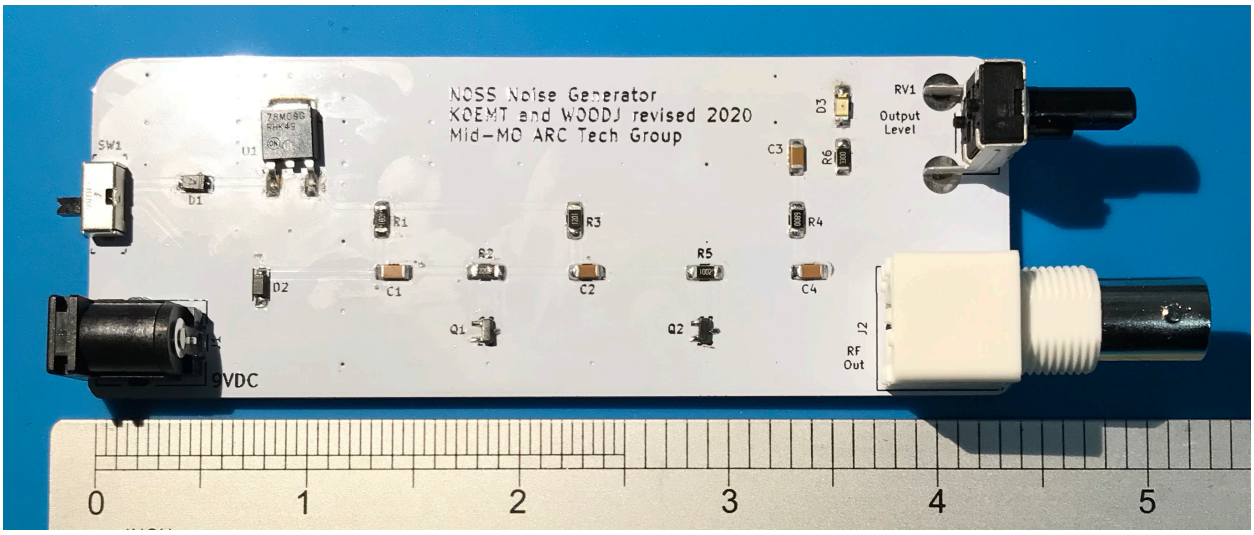
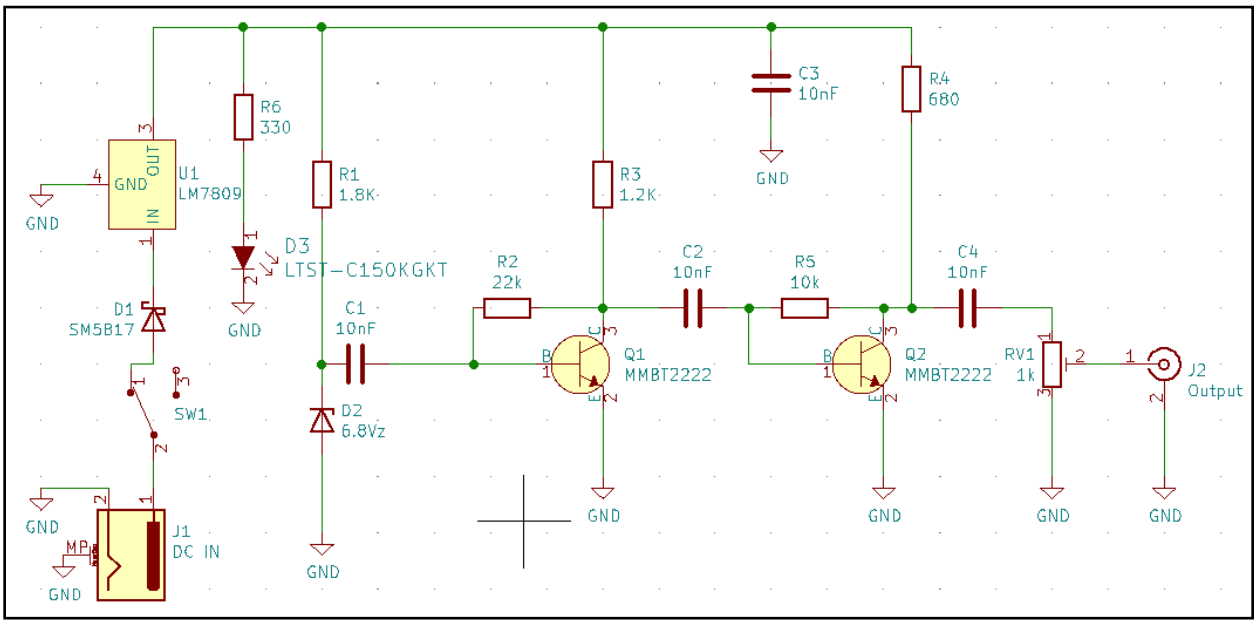


Why this kit?

The Mid-MO Amateur Radio Club Tech Builders group is a community of builders seeking to grow and share their electronics knowledge. This circuit was originally kitted with a THT PCB produced by Tom Hammond, N0SS (SK). In order to further our knowledge of kiCAD circuit and board design as well as continue the N0SS legacy, we undertook the modernization of this noise generator circuit.

The SMD parts are 1206 size. We selected this part size in order to hopefully be a more gentle introduction to SMD kit construction for the radio amateur. Once, you move larger than 1206 in size, part selection and availability becomes more limited. Parts are widely spaced on the board to make building easier for the new SMD builder.



Schematic for N0SS Noise Generator — SMD Adaptation

Parts List

Installed	Footprint	Part	Marking
	C1, C2, C3, C4	10nF	Unmarked, beige
	D1	SM5817	K2
	D2	6.8Vz	E5
	D3	LED	Clear
	J1	DC in	Barrel jack
	J2	BNC	
	Q1, Q2	2222	
	R1	1.8K	1801
	R2	22K	2202
	R3	1.2K	1201
	R4	680	6800
	R5	10K	1002
	R6	330	3300
	RV1	1K	THT Potentiometer
	SW1	SPDT switch	
	U1	LM7809	

Tools

Tools and supplies that you may find helpful:

Fine solder with flux. Soldering iron with a fine tip (wedge or bevel tip with relief for drag soldering.) Hot air station with chip quick solder paste. Ceramic tipped ESD safe tweezer. A “doofus” for part holding. Solder wick. Magnifying device.

Caution: make sure your workbench is safe for working with heat. I used a silicon mat below this board when building.

K0EMT has set up a workbench webpage with links to tools and supplies listed above.

Construction

Phase 1 - Power supply

Taking a cue from K7QO, let's build and test the power supply of the circuit first.

__ Install D1 — bars on the device go into the silk screened U shape. See Figure 1. You will probably need to hold this device down as you solder it.
__ Install R6, 3300
__ Install D3 the LED — on the outer metal contacts, the end with a black U next to the clear material (cathode) should be installed towards the top of the board. See Figure 2.
__ Install U1 the 9V regulator
__ Install SW1
__ The ground lug on J1 is a little too large for the footprint hole. Slightly trim the outside edge of the ground lug. Test fit it. Repeat until the jack can be seated.

Connect to a DC power source. 10.7 VDC is required to produce a regulated 9VDC output. Toggle the power switch, the LED should toggle on/off. Using the open pad of the BNC connector for ground, check that you are seeing ~9VDC from the lead of U1that is nearest the middle of the board.

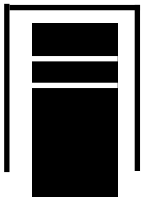


Figure 1: Diode in Footprint

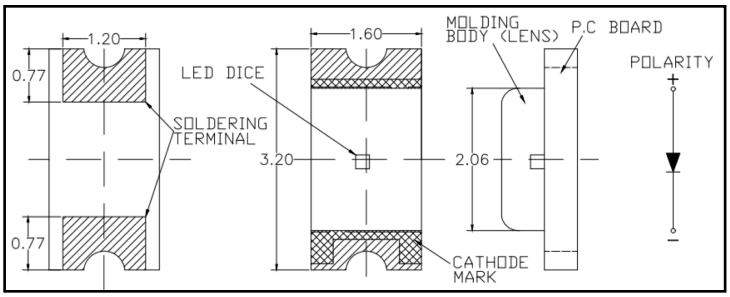


Figure 2: From LiteOn LED data sheet

Phase 2 - Upper portion of board

__ Install C3, beige
__ Install R1, 1801
__ Install R3, 1201
__ Install R4, 6800

Power up the board. Test the pads of C1, C2, and C4 that are directly below R1, R3 and R4 for ~9V.

Phase 3 - Midline board

__ Install D2, labeled E5, bars marking go into U shaped outline, towards top of board
__ Install C1, beige
__ Install R2, 2202
__ Install C2, beige
__ Install R5, 1002
__ Install C4, beige

Here we are checking that everything is in place properly.

__ Use ohm meter to check the top left pad and the middle pad of Q1. Should be same resistance as R2, 22K.
__ Q1, verify no continuity between either of those pads and the bottom pad.
__ Use ohm meter to check the top left pad and the middle pad of Q2. Should be same resistance as R5, 10K.
__ Q2, verify no continuity between either of those pads and the bottom pad.

Phase 4 - Transistors

__ Install Q1
__ Install Q2

Repeating the tests from above, making sure we didn't short the transistors and that they are properly placed.

__ Use ohm meter to check the top left pad and the middle pad of Q1. Should be same resistance as R2, 22K.
__ Q1, verify no continuity between either of those pads and the bottom pad.
__ Use ohm meter to check the top left pad and the middle pad of Q2. Should be same resistance as R5, 10K.
__ Q2, verify no continuity between either of those pads and the bottom pad.

Phase 5 - THT Output

__ Install RV1, 1K potentiometer
__ Install J2, BNC

When the noise generator is turned on you should see 1V DC output from the BNC with RV1 wide open.

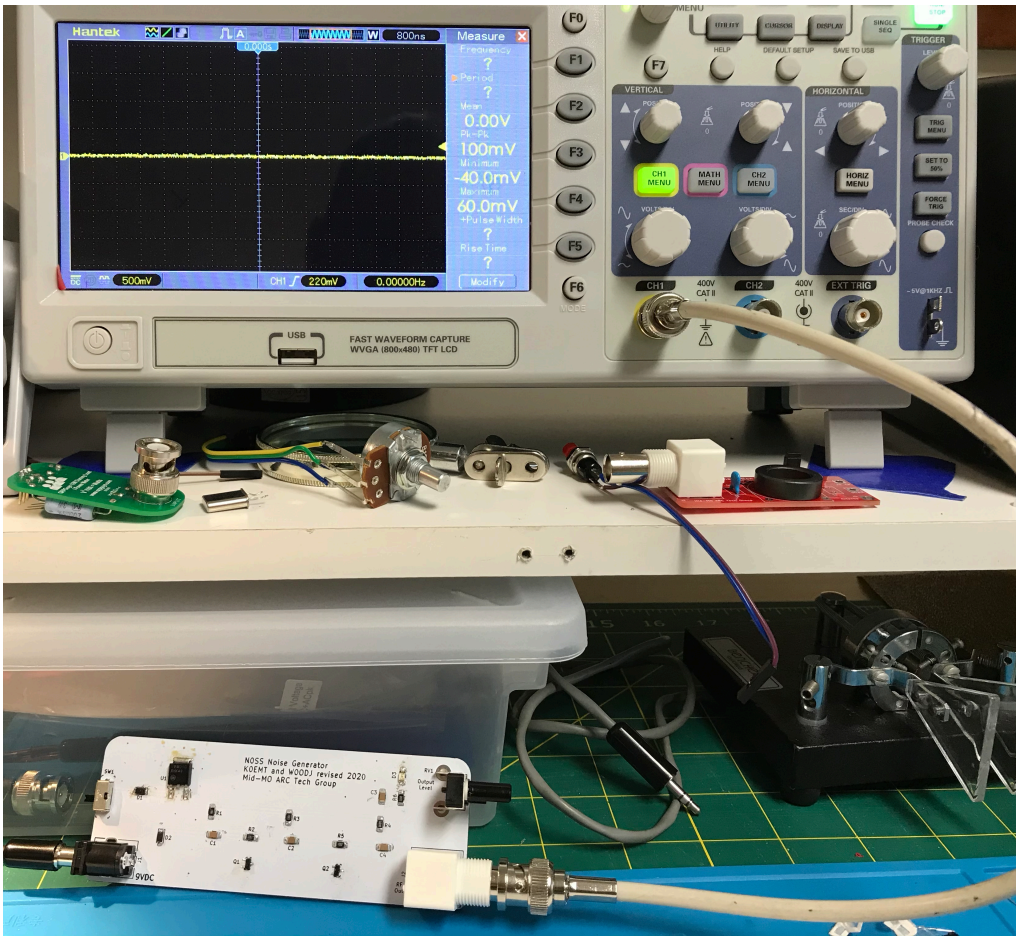
Final Testing

Plug the noise generator into something like the QRPGuys Dummy Load with a flying lead from the RF pin. Tune in a station on your AM radio. Turn on the noise generator. You should hear lots of noise.

If you have a frequency counter that has a very fast gate you may see the frequency jumping around. A 1 second gate will show the frequency changing. But, it won't wildly vary as you might hope for.

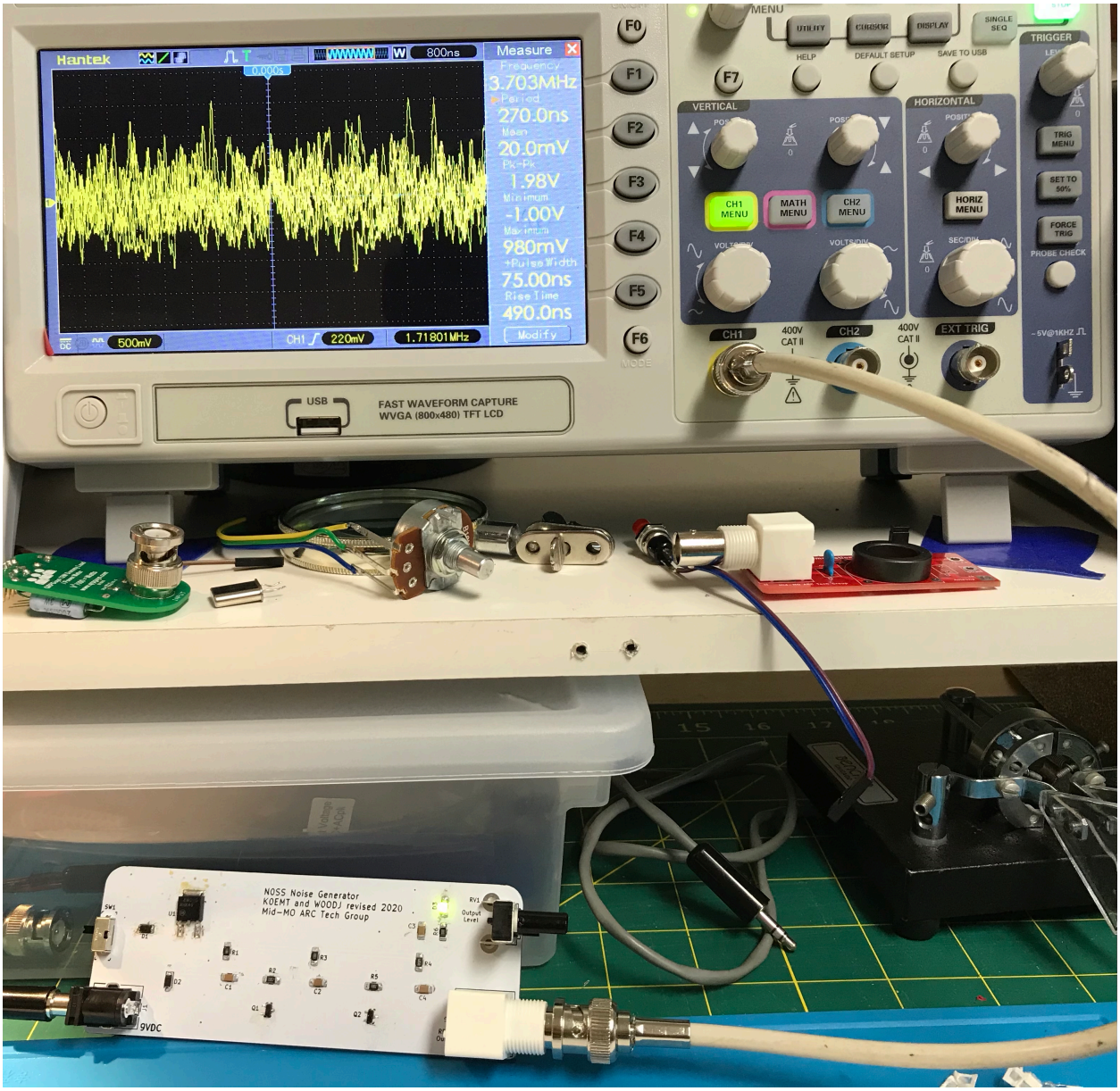
Observing output on an Oscilloscope

Before powering on the noise generator, start with the output adjustment in the fully clockwise position — minimum output. Gradually increase output until your scope is reading it. An output level of ~1V was observed.



Noise generator turned OFF - flat line observed on scope.

Before powering on the noise generator, start with the output adjustment in the fully clockwise position — minimum output. Gradually increase output until your scope is reading it. An output level of ~1V was observed.



Noise Generator in operation

Spectrum Analysis

Operation

There was a mental error made when labeling the DC jack. With the prototype, a minimum of 10.7V DC input was required to get 9V regulation. The MC78M09 device is rated for a 18V maximum input. Let's stick with ~12V.