A noise Power Indicator ... by Brian Coleman, G4NNS

A noise power indicator can be a useful tool for judging the performance of microwave receivers using the Y factor method (cold sky vs warm object). This relies on the fact that all bodies above absolute zero emit “black body” radiation which can be detected at the higher microwave frequencies. In practice this means that you can use the “cold” sky vs a known “warm” reference. I find aiming the antenna at the house nearby is best and you can assume its temperature is about 293K. Measurement as opposed to indication can be accomplished by using a precision attenuator in front of the indicator and using the same level at the detector to avoid any non-linearity in the system. I also use my system to ensure that my EME antenna is tracking correctly using moon noise of which I see about 2.5dB at 10GHz. Other uses of this tool can include antenna testing. It provides a more precise and easier to read system than an S-meter and can, in conjunction with a precision attenuator provide a means of making fairly precise measurements of the order of +/- 0.2dB. When using a warm source such as your house remember that it should fill the aperture of your antenna. This is why small antennas (more than about 0.5 degrees beam width) do not see the full potential sun or moon noise.

Circuit description (see Figure 1)

The system consists of a high gain amplifier (circa 70dBs) and a sensitive detector at the IF frequency (in this case 144MHz). The bandwidth needs to be as wide as possible but not exceeding that of the transverter front end. This is important as noise generated outside the desired pass band of the transverter is not relevant to the system performance and could swamp the power indication and lead to misleadingly poor indication of receiver performance. The 3dB bandwidth of this indicator is about 1MHz. Relative measurements are made by using a switched attenuator in front of the noise power indicator. Such attenuators can be found at microwave flea markets.

The main challenges to the circuit design and construction are:- 1) minimising the gain required to drive the detector. I.e. the detector must be as sensitive as possible and 2) keeping the gain stages stable at all frequencies including those outside the intended pass band. The first requirement is met by using forward bias on the schottky detector diode to move it into its linear and most sensitive region and by using a simple op-amp to amplify the DC difference between the RF detector and another matched reference diode, similarly forward biased but decoupled to RF, in the same SM package. The second requirement is achieved by using MMIC gain stages with careful construction, grounding and shielding. With so much gain on one frequency, great care has to be taken in the layout and construction to avoid instability. The stage before the detector and second band pass filter uses a JFET as this is more tolerant of impedance mis-match. No originality is claimed for any part of the circuit but some time and effort has been spent in trying to ensure the repeatability of the layout and simplicity of the PCB. If good practice for RF construction is followed, it should be possible to build the circuit and get it to be stable quite easily. The input must however look substantially resistive and be correctly matched at all frequencies. This can be achieved by the use of some fixed attenuation at the input.

Construction (see Figure 2)

A simple way to make the PCB is to cover the double sided copper laminate with sticky tape such as 3M Magic tape then stick a print of the track layout onto the tape and use a scalpel or sharp modelling knife to cut out and peel off the areas to be exposed to the etchant.

The design uses as many straight lines as possible to simplify this. I found it helpful to use a straight edge under a 3D magnifier. After etching and cleaning the PCB, the screens should be cut to size. The long, side screens are best made using 1/16” laminate as used for the main PCB. The inter stage screens are best made from 1/32” copper laminate or from tin plate. I did not find end screens to be necessary but if you fit them they should be made from 1/16” laminate. Drill 2.5mm holes to recess the MAR8s, which should be the “drop in” type, to minimise lead lengths. Also drill 0.8mm holes for the Via pins. These pins are essential for stability and it is possible that more than the four shown may be needed. (see setting up).

As with any PCB construction it is best to start with the smallest components first as the larger ones will otherwise get in the way **(see figures 3 & 4)**. Whilst the screens should be fitted last you may find it easier to fit L1 and L2 after fitting the long side screen closest to these inductors as this will make it easier to solder the seam. L1 and L2 are separated by about 3mm and are 2-3mm above the ground plane. Note also that the long side screens extend below the main PCB and should match the height of the inter-stage screens on the component side. They should be soldered both to the top and bottom surfaces of the main PCB. Make sure the side screens remain at 90 degrees to the PCB by “tack soldering” both sides and repositioning as necessary before soldering the entire length of the seams.

When assembling the DC amplifier it is again best to fit the smallest components including the wire links and pins first. I find it best to make the track cuts after the components are in place and to use this process as a check for correct placement of the components. Start fitting the active components with the +5V regulator, and test this before fitting the voltage converter. Test that the + and – 10V rails are working before fitting the op-amp.

Setting up

Terminate the input with a good 50 Ohm load. This is essential for stability. Adjust VR1 for zero reading on the meter. If this is not possible the amplifier may be oscillating. This should not be happening! If it is try touching different gain stages to see if the meter reading changes. You may be able to identify which stage is oscillating by this method. You could try lifting one end of each one Ohm resistor in turn to identify the oscillating stage. Whatever you try, make sure it is reversible and restore it to the original condition before trying something else. Here are some things to try if necessary: 1) Change the capacitor and inductor tuning to see if the oscillation stops. 2) Add additional via pins close to the MAR8s. Those indicated in the design have proved sufficient but in some cases more may be required to achieve stability. 3) Try adjusting the values for R1, R6 or R7 for the stage that is oscillating.

Once the amplifier is stable and the meter set to zero, apply a small signal within the pass-band e.g. 144.50MHz from a 50 Ohm source through a switched attenuator. Tune VC1, VC2, L3 and L4 for maximum reading increasing the attenuation as you go to keep the meter on scale. The prototypes, when set up as described, had a Full Scale reading on the most sensitive range, corresponding to about –80 to –85dBm.

Next, connect to the receiver or transverter under test. The Indicator should show a non zero reading when the transverter is switched on but make sure there are no strong signals in the pass-band, such as local beacons. If the meter is on the end stop try inserting some attenuation between the transverter and the indicator

Applications

Receiver performance can be measured using ground noise and quiet sky noise. For EME systems where the antenna beam width is close to or less than the angular diameter of the source, Sun and moon noise measurements provide a useful indication of receive performance. The indicator can also be used for calibration tracking systems. The indicator also has potential for plotting polar diagrams and measuring gain of antennas for 2m or bands where a transverter has a 2m IF.

For EME, the indicator can display moon noise during receive periods to provide confirmation that the tracking system is functioning correctly.

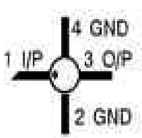
For this purpose, it is necessary to supply a sample of the IF to the indicator through a splitter (T) and to switch the indicator out of circuit during transmit periods. It is desirable to switch off some of gain stages during transmit periods, to avoid overload. Some gain drift may be experienced but this is minimised if the smallest number of stages are switched off. Try switching off just the first one or two stages if possible. A spare power track is supplied on the PCB for this purpose. The circuit diagram shows the possible arrangement for RF switching. The number of stages to be switched will depend on the isolation of the relay used.

Notes

1. If you fit the unit in a box and it becomes unstable, try lining the box with conducting foam or lossy rubber where it is closest to the component side of the Noise amplifier.
2. On the supplied veroboard component layout for the DC amplifier, **C26** is missing. This decoupling capacitor can be conveniently soldered across the MX680 IC, from pin 8 to pin 5.
3. Further information, if required can be obtained from Brian, G4NNS, QTHR or

   email to: brian-coleman@tiscali.co.uk

4. A component list, **figure 4**, is provided at the end of this article.
5. Except for the MAR8 modamps, semiconductor pin outs are shown on the circuit diagram.



MAR8 pin out connections

Figure 1

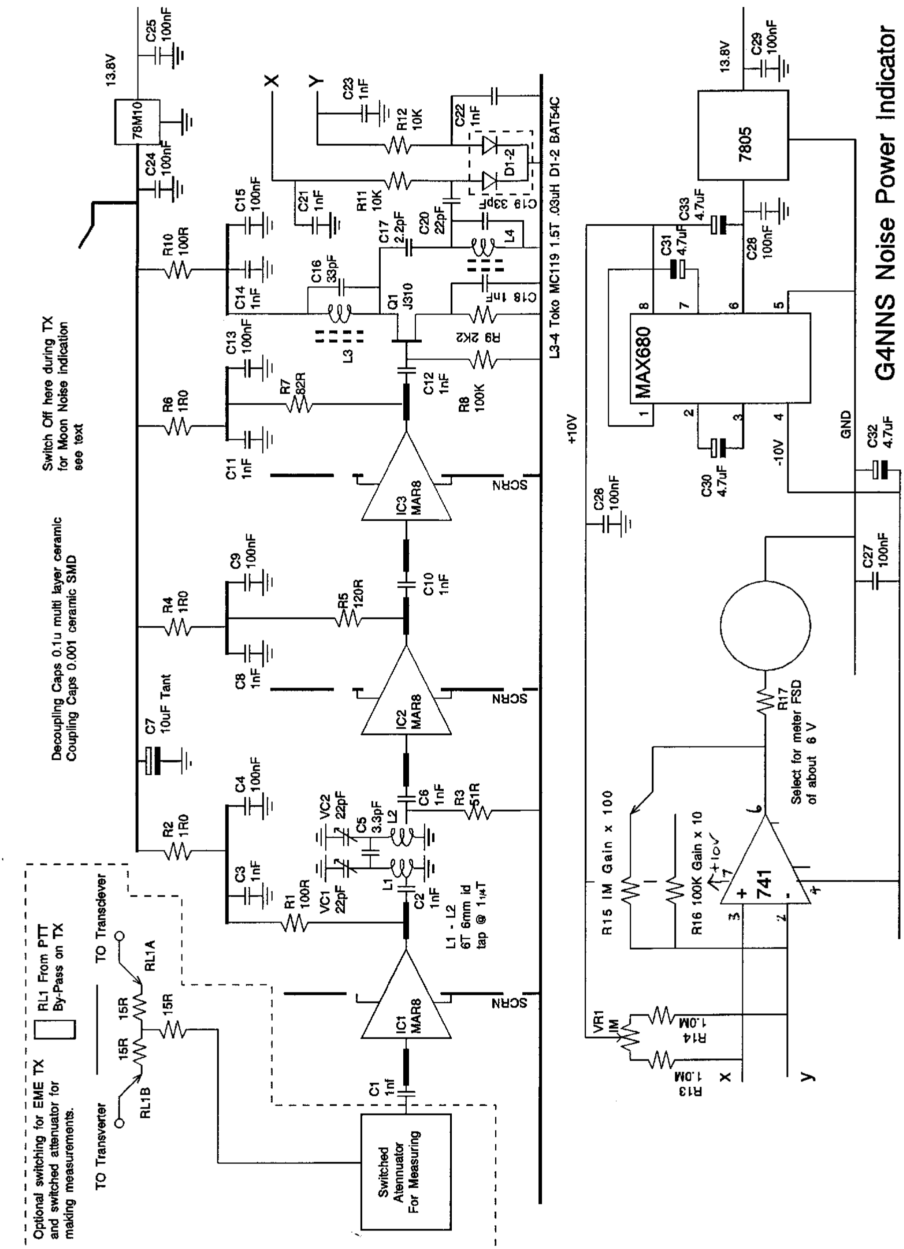
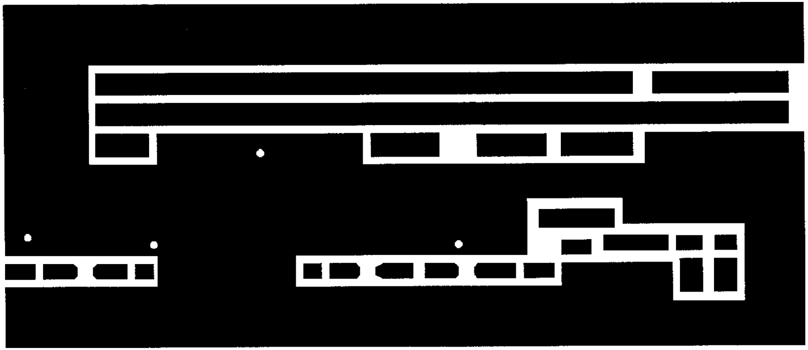

G4NNS Noise Power Indicator
Decoupling Caps 0.1u multi layer ceramic
Coupling Caps 0.001 ceramic SMD
Switch Off here during TX
for Moon Noise indication
see text
Optional switching for EME TX
and switched attenuator for
making measurements.
RL1 From PTT
By-Pass on TX
TO Transverter
TO Transciever
RL1A
RL1B
15R
Switched
Atennuator
For Measuring
C1
1nf
IC1
MAR8
IC2
MAR8
IC3
MAR8
SCRN
L1 - L2
6T 6mm id
tap @ 1¼T
L3-4 Toko MC119 1.5T .03uH D1-2 BAT54C
Q1
J310
13.8V
78M10
7805
MAX680
741
Select for meter FSD
of about 6 V
R15 1M Gain x 100
R16 100K Gain x 10
VR1
1M
R13
1.0M
R14
1.0M
+10V
-10V
GND
X
Y

Figure 2

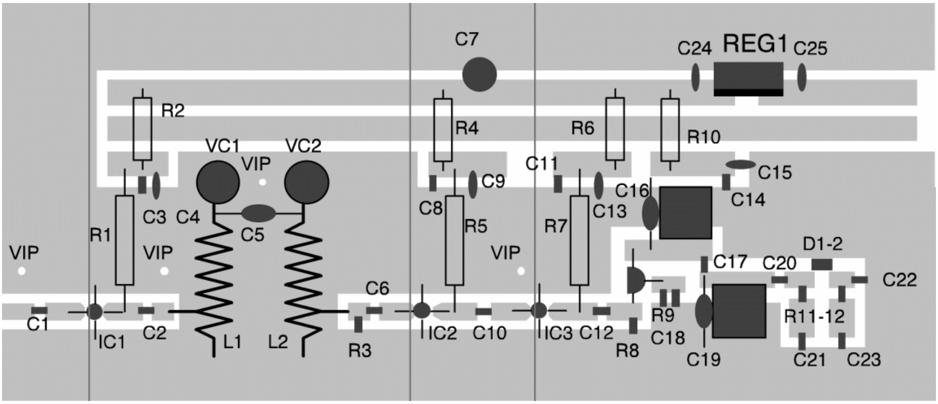


Main PCB 1/16 Laminate 105 x 45mm
Note position of 4 via holes.

Intermediate Screens
1/32" Laminate
45 x 20mm
QTY 3

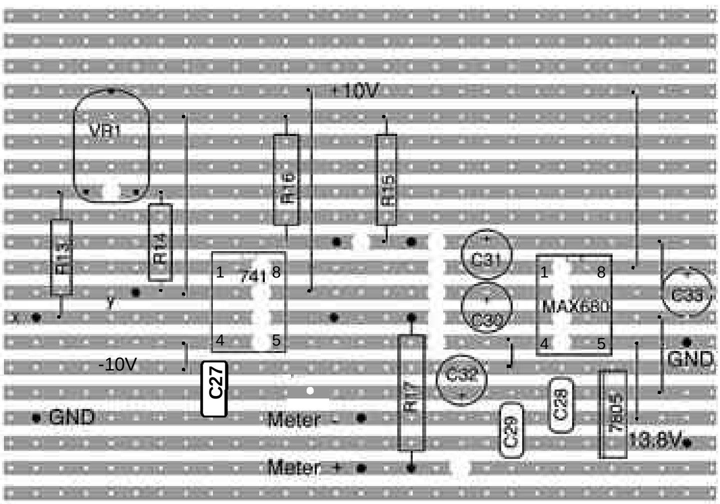
Side screens 1/16" Laminate 105 x 25mm . . . QTY 2

G4NNS NOISE AMPLIFIER (continued)



Above: Main amplifier pcb tack and component layout

Below: DC amplifier Veroboard layout. Note that C26 is missing from this diagram and should be soldered from the +10V line to ground at a convenient point (eg across pins 5 and 8 of the MAX680)



G4NNS NOISE AMPLIFIER—COMPONENT LIST

R1	100R Ax	
R2	1R Ax	
R3	51R SMD	
R4	1R Ax	
R5	120R Ax	
R6	1R Ax	
R7	82R Ax	
R8	100K SM	
R9	2K2 SM	
R10	100R Ax	
R11	10K SM	
R12	10K SM	
R13	1M0 Ax	
R14	1M0 Ax	
VR1	1M	
R15	1M Ax	
R16	100K Ax	
R17	SOT	
VR1	1M0	
VC1	22pF	
VC2	22pF	
L1	6T 6mm id	
L2	6T 6mm id	
L3	MC119 1.5T	1st 10 SAE to G4NNS
L4	MC119 1.5T	or try BONEX
Q1	J310	
IC4	741	
IC5	MAX 680	Farnell 246_451
REG1	7810	
REG2	7805	
IC1	MAR8	
IC2	MAR8	
IC3	MAR8	
D1-2	BAT 54C	Farnell 302_284

C1	1nF SM
C2	1nF SM
C3	1nF SM
C4	100nF SM
C5	3.3pF Rad
C6	1nF SM
C7	10uf Tant
C8	1nF SM
C9	100nF SM
C10	1nF SM
C11	1nF SM
C12	1nF SM
C13	100nF SM
C14	1nF SM
C15	100nF SM
C16	33pf Rad
C17	2.2pF SM
C18	1nF SM
C19	33pF Rad
C20	22pF SM
C21	1nF SM
C22	1nF SM
C23	1nF SM
C24	100nF SM
C25	100nF SM
C26	100nF Rad
C27	100nF Rad
C28	100nF Rad
C29	100nF Rad
C30	4.7 uF
C31	4.7 uF
C32	4.7 uF
C33	4.7 uF