QRP Labs 20W 50Ω Dummy Load Kit Build

Recently, I was looking for buildable project ideas for a soldering class to be held at the GCARC Clubhouse, and I came across a low-cost 20W 50Ω air-cooled dummy load (Figure 1) in a very small form factor, perfectly suitable for use with the (tr)uSDX radio or others of its kind. I ordered in the kit and decided to put it together. That experience led to this article.

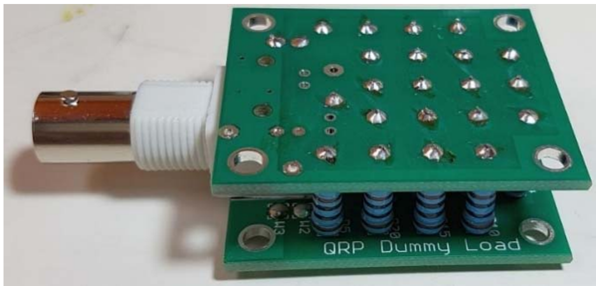

Figure 1 - QRP Labs Dummy Load (photo courtesy of QRP Labs)

The kit comes from *QRP Labs* (https://qrp-labs.com/) at a relatively low cost of $8.50 (USD) plus shipping. Two shipping options are offered – FedEx/TNT at $14.41 or standard postal shipping at $8.53, both of these figures being in US dollars. I mention that because when it arrived, I saw that the kit had been shipped from Turkey (Figure 2).

Figure 2 - Package from Turkey

The kit arrived in less than two weeks from the date of order, and it arrived in good condition. The funny thing is that the *QRP Labs* website offers a warning that the postal shipping may be slow. I figure two weeks from Turkey to be average for overseas non-priority shipping, so I am OK with the two weeks transit time.

In the kit package were a pair of identical printed circuit boards still attached at their join line, a tape of twenty-two 1kΩ 1% one-watt resistors, a BNC connector, a 1N4004 diode, and a 10nF monolithic capacitor (Figure 3). The two PCB's, as I mentioned, are identical… and are used in sandwich form with the resistors placed in between the boards.

Figure 3 - Kit contents

Assembly of the unit starts out with the separation of the two PCB's, and then the marking of one particular hole on one of the two PCB's. Locate the hole marked *"W0"* on one of the PCB;s, and mark that hole on the non-screen-printed side of that PCB, which will then become the "upper" PCB in the assembled sandwich. Marking of the hole can be done with a fine-point Sharpie® marker. We will be using that marked hole later on in the build process, when we will need to locate the hole without being able to see the screen-printing on the PCB. An alternative, and possibly better (so long as no errors are made) approach is to solder-fill hole *"W0"* in the PCB that will become the lower board, and to solder-fill hole *"W1"* in the PCB that will become the upper board.

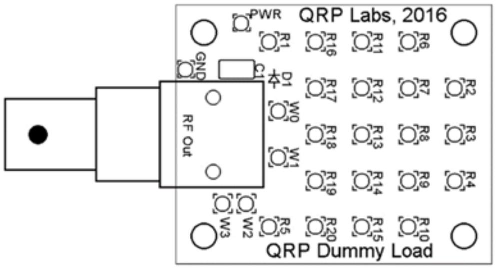

Figure 4 - PCB drawing

Now it is time for the installation of the 1N4004 diode and the 10nF capacitor to the screen-printed side of the unmarked one of the two PCB's (the one determined to be the lower board if solder-filling was done as described above), in the locations indicated on the boards. The diode polarity is not really clearly marked because the marking is crammed between two holes in the PCB. As it turns out, the instruction manual for the assembly does clearly show the diode orientation in the image of the PCB (Figure 4). This diode is installed into those two holes, on end, with the holes spaced 0.100" apart. When bending the diode to fit the board, bend the anode lead (the end without the band) and then insert the cathode (banded) end into the hole directly adjacent to the "D" in the *D1* location identifier screen-printed on the PCB. The anode lead will then go into the hole directly adjacent to the "1" in the screen-printed *D1* identifier.

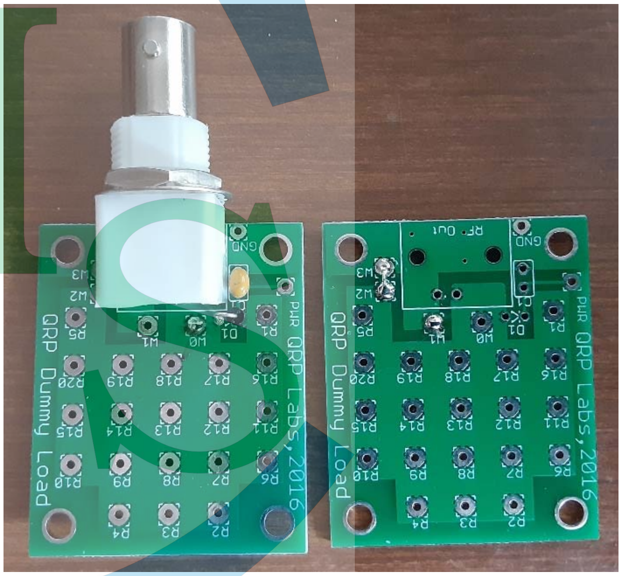

Figure 5 - PCB's ready for resistors

A quick word about the instruction manual. It is *not* included with the kit as shipped, apparently in a bid to keep the shipping costs as low as possible. However, the manual is readily available on the *QRP Labs* web page where the kit is described (https://qrp-labs.com/dummy.html). These guys are pretty good. They offer the instruction manual in English, French, Czech, and Japanese. Take your pick.

After the diode and capacitor are in place, solder the BNC connector into place on the screen-printed side of the lower PCB. Next, a small jumper wire must be installed on the opposite PCB, which will become the upper board in the sandwich. A cut-off lead from the diode will work just fine for this jumper wire, which gets installed between the holes marked W2 and W3 on the screen-printed side of the upper PCB. Remember, this is the opposite PCB from the one that got the diode and capacitor (Figure 5).

Next up is to install twenty resistors in the holes marked R1 through R20 on the lower PCB, inserting them from the screen-printed side, placing them all the way down to the board. These resistors should be carefully installed, so that all of the resistor bodies are parallel to each other physically so as to give the best assembled appearance, as if of soldiers in ranks. I found that the easiest way to do this is to drop the resistors into place in the upper board, which I had secured in my PCB vise. Then, I lowered the lower PCB down onto the resistor leads, aligning them to the proper holes in the upper PCB. After tacking the resistors in place, I lifted the upper

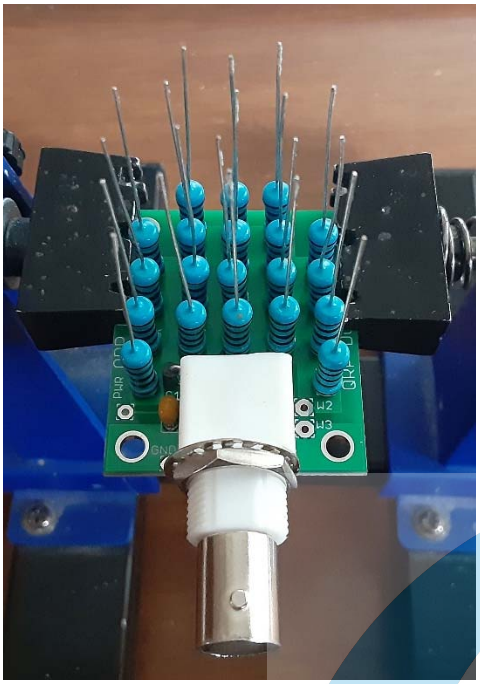

Figure 6 - Resistors installed in lower PCB

PCB off the lower board, and then touched up the positions of each resistor individually, arranging them into proper parallel ranks (Figure 6).

In order to simplify the installation of the resistors' opposite leads into the upper PCB, modify the installed resistors as follows:

- in the far edge row of resistors (R2, R3, & R4), leave the leads full length;
- in the next row of resistors (R6, R7, R8, R9, & R10), cut off about an eighth of an inch of each lead;
- in the third row of resistors (R11, R12, R13, R14, & R15), cut off about one quarter of an inch of each lead;
- in the fourth row of resistors (R16, R16, R18, R19, & R20), cut off about three-eighths of an inch of each lead; and
- in the final row of resistors (R1 and R5), cut the leads off the same as in the fourth row above, removing about three-eighths of an inch of each of those two leads.

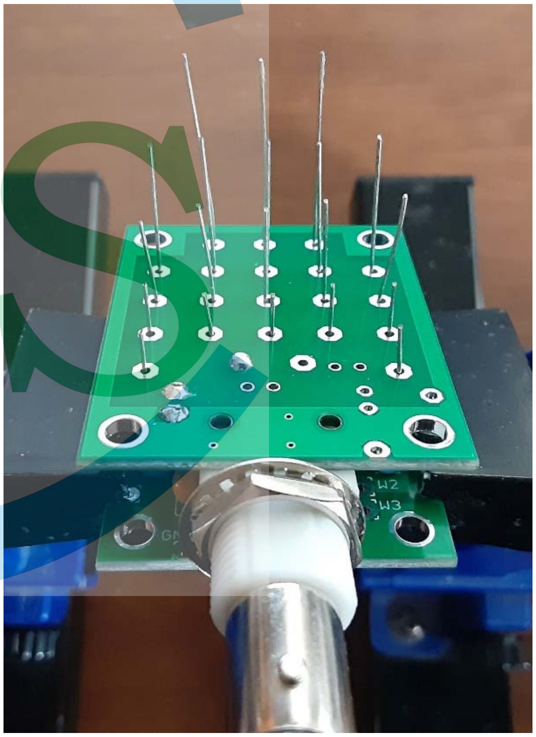

Figure 7 - Staggered lead lengths

Staggering the lengths of the resistors by row in this manner (Figure 7) will make it much easier to align and insert the resistor leads into the upper PCB in the sandwich. Position the upper PCB over the cut-off resistor leads with the screen-printed side down (or towards the resistors) and begin inserting the resistors into the holes in the PCB, one row at a time and starting with the tallest row – the row in which no lead length was cut off. Once all of the resistors are inserted into the upper PCB, lower the PCB down against the upper surface of the BNC connector body, ensuring that the PCB is level. Solder one or two of the resistors to the upper PCB and stop to check the positioning of the upper PCB. Correct the PCB position as required by reheating the solder joint(s) and repositioning the board as needed.

For appearance's sake, it is important that the PCB's are aligned as nearly perfectly as is possible. All that it takes to achieve this alignment is to solder just a few of the resistors at first, adjusting the position and alignment of the upper PCB as you solder each lead. Once the upper PCB is positioned exactly as you want it to be, go ahead and solder the rest of the resistor leads to the upper PCB (Figure 8).

Once all of the resistors are soldered in place and the board is positioned level and in proper alignment to the lower board, it is time to make the final electrical connection of the upper resistor ends to the ground plane of the lower PCB. This is done by inserting a bare wire – one of the full-length cut-off resistor leads works well here – into hole W0 on the upper PCB, which is the hole that we marked with the Sharpie® marker earlier. This bare wire needs to continue straight down through the PCB sandwich until it enters hole W1 in the lower PCB. Solder this bare wire in place.

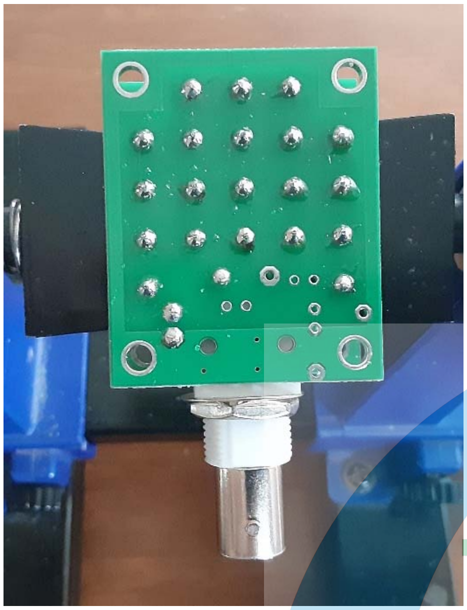
Figure 8 - All resistors soldered and clipped

Note that when the dummy load soldering is complete, hole W0 in the lower PCB and hole W1 in the upper PCB will not be populated. This is by design, and is a result of the desire to use two identical PCB's in the design for simplicity. Of course, if the solder-fill method of PCB marking was used, these holes will be blocked with solder already.

Cut off the excess lead length from any leads not already clipped, making the finished appearance as clean and close as is possible. Then, using a small smooth file, clean up each clipped lead solder mound (Figure 9) to remove any and all peaks, hooks, and snags that may exist, aiming for a nicely smooth finish, as this will be the final exterior surface of the finished device. Repeat this process with the underside of the lower PCB in the sandwich (Figure 10).

I chose to finish my kit with an insulating coating on both boards. I accomplished this by applying three coats of clear nail polish over the entire foil side (Figure 11) of each PCB in turn, allowing the polish to dry completely between coats. Then, using some acetone on a cotton swab, I removed the polish from the PWR and GND hole areas on the lower PCB. You could accomplish the same result using a clear aerosol spray enamel, but you would have to mask the two holes to prevent coating them. These two holes are power measurement points that we will discuss shortly.

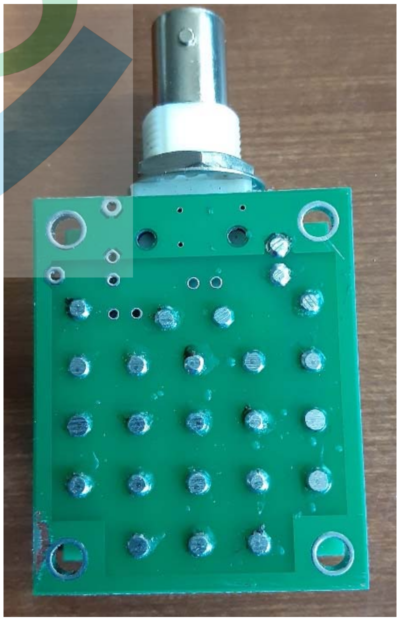
Figure 9 - Upper PCB after file work

Let's take a few minutes to discuss the technical aspects of this kit. As a dummy load, it certainly meets certain performance standards, as previously stated, in that it is a 50Ω load rated for 20 watts RF power. Examining each of these specifications with regard to the device schematic, we can see what we are getting.

Using the familiar reciprocal of the sum of the reciprocals formula for parallel resistance…

$$R_{PARALLEL} = 1 / [(1/R1) + (1/R2) + (1/R3)...]$$

We can now see that what we are working with is…

$$1 / [\,(1/1000) \times 20] \text{ or } 1 / 0.02 = 50\Omega$$

Thus, a total of twenty one-kilohm resistors taken in parallel will produce a net resistance of fifty ohms (50Ω). Of course, the resistor tolerance must be factored in when determining the actual final impedance offered by these twenty resistors in parallel. In this case, the resistors are 1% tolerance devices, which means that each resistor should be somewhere between 990Ω and 1010Ω to remain within the stated tolerance. Thus, if *all* of the

Figure 10 - Lower PCB after filing

resistors were at the low end of their tolerance range, we would have a calculated impedance of 49.5Ω offered, and if *all* of the resistors were at the high end of their tolerance range, the calculated impedance offered will be 50.5Ω. The reality is that it will fall somewhere between these two extremes, and as such, it will be close enough to the 50Ω characteristic output impedance of our radios to be acceptable.

OK – so much for the impedance. How about the power handling capability? That is resolved in a simple and straightforward manner. Because all of the resistors in the parallel group are of the same power rating, we can simply multiply that power rating by the number of resistors involved. In this case, we are using one-watt resistors, of which there are twenty in use. Therefore, the power handling capability of the finished dummy load is $20\ x\ 1W = 20W.$

What about the diode and capacitor in this device? What purpose(s) do they serve? In essence, they are the heart of a simple RF detector that allows us to make some rudimentary (non-precision) power measurements from the dummy load. The diode/capacitor combination rectifies and smooths a sample of the RF power applied, enabling us to get a relative power level reading using a standard analog or digital multimeter.

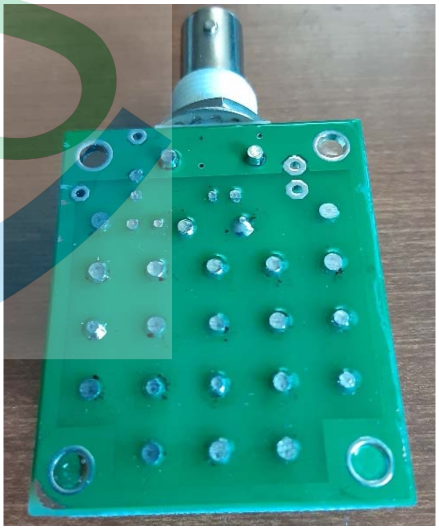

Figure 11 - PCB with clear polish coating

The PCB’s each have a pair of holes labeled PWR and GND (Figure 12). It is between these two points, *on the lower PCB only,* that we can make the relative power measurement. These are the holes that we avoided insulating earlier when we coated the PCB’s with clear nail polish or clear spray enamel. This measurement will be a peak voltage measurement, which is converted to a power reading using a variant of the basic formula

$$P = V^2 / R$$

where P is the calculated power, V is the measured voltage, and R is the impedance or resistance of the active circuit. To this end, there is what might appear to be an error in the kit manual, wherein the manual states that the formula to be used is

$$\text{“ Power} = \text{peak voltage} * \text{peak voltage} / 100 \text{ ”}$$

which uses 100 ohms as the resistance or impedance value, double the actual impedance of the dummy load design. As a result, the calculated power ends up being one half of what one might expect it to be if using the expected value of fifty ohms (50Ω) in the equation. What this means is that the example 20V peak voltage measurement would yield a calculated power level of 8 watts instead of the correctly stated 4 watts.

However, it must be remembered that we are dealing with RF voltage, which in the case of an unmodulated carrier is a sine wave, and the voltage measurement made is actually a *peak* voltage (V_P) reading instead of the more-commonly measured V_{RMS} as read on most multimeters. This changes the arithmetic somewhat, as follows.

While the basic formula stays the same as $P = V^2 / R$, the voltage component changes. What we are left with in this case is the fact that at RF, the formula becomes $P_{RF} = V_{RMS}^2 / R$. As you may recall if you are a General or an Amateur Extra ham operator, $V_{RMS} = V_P \text{ x } 0.707$. Arithmetically, the value 0.707 is equivalent to $1/\sqrt{2}$, so the formula can be re-written as $V_{RMS} = V_P \text{ x } 1/\sqrt{2}$, or more simply as $V_{RMS} = V_P / \sqrt{2}$. Stay with me here… it is going to get interesting.

Using the above, we can now re-write the original formula as:

$POWER_{RF} = (V_P / \sqrt{2})^2 / R$, or as

$$POWER_{RF} = (V_P \text{ x } (1 / \sqrt{2})) \text{ x } (V_P \text{ x } (1 / \sqrt{2})) / R.$$

Let's consolidate and then get rid of the two instances of "one over the square root of two" from the equation…

$POWER_{RF} = V_P \text{ x } V_P / R \text{ x } \sqrt{2} \text{ x } \sqrt{2}$ which is equivalent to

$POWER_{RF} = V_P \text{ x } V_P / R \text{ x } 2$ or $POWER_{RF} = V_P^2 / 2R$.

If R is fifty ohms as it is in the dummy load, the net result is $POWER_{RF} = V_P^2 / 100$.

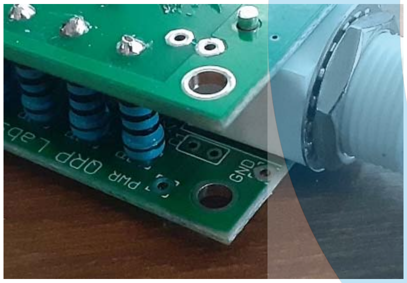

Figure 12 - PWR and GND holes

The manual correctly states that this is *not* a precise quantitative measurement of the power, but instead is an approximate or relative power level. In order to derive an accurate power measurement, the forward voltage drop of approximately 0.6V to 0.7V must be taken into consideration. In addition, the diode's forward current, which in turn is a function of the DVM input impedance, will also affect the specific power measurement accuracy. Finally, there is a frequency dependency that must be taken into account, which is a result of the RC filtering that occurs because of the series capacitor in the circuit.

The long and short of all of this is that while the PWR and GND points on the lower PCB provide convenient test points for making peak voltage measurements, these measurements will only roughly translate to an RF output power of the transmitter to which the dummy load is connected. The instruction manual provides a couple of performance graphs for the dummy load, which depict the performance of what one must consider to be a "typical" example of this dummy load. A 10MHz RF signal was input to the dummy load, with its power varied form 0.8mW all the way up to 10 watts. The input signal was passed through a 30m low-pass filter before entering the dummy load. An inexpensive *"Harbor Freight Tools"* type of digital volt meter was used to take the actual voltage measurements, while the actual measured power values were derived through the use of a 100MHz bandwidth digital oscilloscope in its *peak-peak* measurement mode.

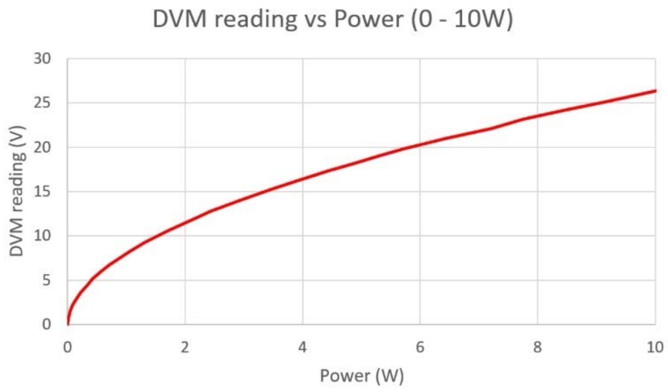

Figure 13 - Zero to 10 watts graph (courtesy of QRP Labs)

The graph at Figure 13 shows the entire power spread all the way up to the 10W input level. What is graphed are the actual measured power levels validated via the digital oscilloscope, plotted against the actual voltage measurements obtained at the PWR and GND points on the lower PCB.

It should be noted that the results shown in this graph do not quite track the anticipated results from the formula discussed earlier. For example, the manual stated that a 20V reading would equate to 4 watts. Instead, the graph shows just about 6 watts against the 20V measurement.

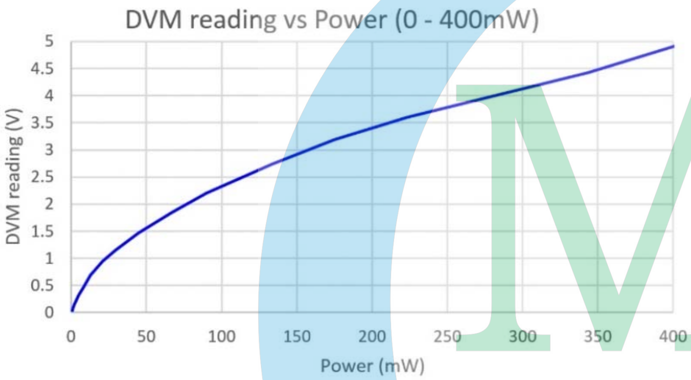

Figure 14 - First 400mW of Figure 13 graph (courtesy of QRP Labs)

The second graph provided, shown at Figure 14, is a close-up view of the first 400mW segment of the Figure 13 graph. Because of the zoomed nature of this graph, we should be able to see a bit more accurately just what the voltage/power relationship is through this device. If we apply the accepted standard formula for power from voltage and resistance ($P_{RF} = V_P^2 / 2R$), we would see that a 3V measured voltage should equate to 90mW ($(3 \text{ x } 3) / 100 = 0.090$). However, on the Figure 14 graph, the 3V measurement indexes to considerably more, coming in at an extrapolated 155-160 mW. That value is not close enough that I can accept the effects of the diode and capacitor as the reasons for the difference. My further problem is that the difference seems to grow all out of proportion as the power levels go up.

In any event, this dummy load is not intended to be an *accurate* source of power measurement. That much is understood. The question is *"How much error is acceptable before the power measurement is no longer relevant?"* The answer is, to me at least, that when the error becomes greater than about 10%, it is no longer relevant and should be ignored.

This kit, as a finished product (Figure 15), does what it is supposed to do. After all, it is marketed as a dummy load, not as a power measurement adapter, right? As such, I have to give the kit some good marks, though there are some criticisms.

I am not pleased by the fact that the BNC jack is supported only by the two signal path leads – the signal lead and the ground lead. The jack has two locating pins which fit into holes in the PCB. These two holes should have been made as pads to which the support pins could be soldered. As it is now, the jack has some wiggle room between the PCB's, meaning that each time the jack is wiggled, the two signal leads are being flexed. Eventually, one or both of them will break off. It would have been an easy change to the PCB design, which would not have cost any more in the board production costs, to have made these holes into solderable pads. They already penetrate the ground plane. All that was needed was to exclude the solder mask around the holes. This is an ugly oversight, and it needs to be corrected in future board production runs.

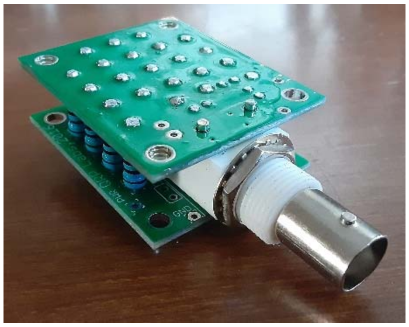
Figure 15 - Finished product

The second criticism is the fact that there were twenty-two resistors included in the kit. While this is a very minor point, it seems to me that this is poor cost control. Consider this… for every ten kits packed with two extra resistors, they have over-shipped enough resistors for another kit. The stated reason for the extra resistors is in case of "shipping errors". I would be more concerned about the little things like the diode and the capacitor before I worried about missing two resistors in a taped set of twenty, especially when one considers that the taped resistors can be cut accurately by length, without the need to actually count each resistor. While the price for this kit is quite reasonable, it could be even more so if the kit supplier shipped twenty resistors instead of twenty-two in each kit.

While the instruction manual has a truncated schematic diagram of the dummy load, I prefer working with complete schematics. To that end, I drew up a schematic that represents all of the components and jumper wires of this kit. Is it necessary? Probably not… but then I do like to be thorough and complete in these things. The schematic diagram is shown in Figure 16.

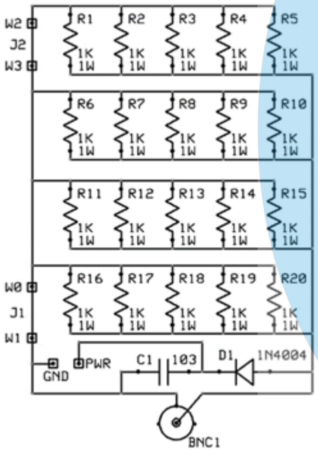

Figure 16 - Schematic diagram

For ease of kit assembly, this kit gets a B+. The things that bring it down from being an "A" are the fact that the whole deal with the two identical boards can be confusing to some builders. In addition, the diode placement is a bit ambiguous to the less experienced builder. Finally, the assembly manual does not give any ideas or assistance when it comes to marking the *"W0"* hole in the upper PCB, leaving the builder to figure that one out alone. This is not a problem for an experienced kit builder, but is probably confusing for a beginner. In spite of all of that, I would still consider this to be a kit that is suitable for kit-building beginners.

The finished product is a nice compact dummy load that is small and light enough to be attached directly to a BNC antenna jack on a QRP radio, using a dual male BNC adapter is necessary to change genders. This makes it useful for POTA and other backpack-type radio operations where a dummy load might be needed for off-air tuning of a radio. If you want or need a small, lightweight and easily deployed dummy load for your QRP work, you should give this one a look. All things considered, this kit merits a place in any shack where QRP comms are a part of the operator's repertoire. There might be better, or cheaper, or easier to assemble dummy loads out there… but this one covers all of the bases and checks off all of the boxes. It gets a considered approval from me both for its design, its cost, and its functionality.