After the joints have cooled, inspect them carefully to make sure that they are solid and make contact with the board. If you have any doubt about the quality of the connections as in the picture not enough solder, apply more flux and reflow the joints until you are satisfied.

Once you are happy with the joint follow the process for R2 – 40, measure the resistance and it should be 50 Ohm or close to it (within 5%). Now solder R41 (4.7M ohm), C1 (10nF) and D1 (1N5711) in posistions marked on the PCB. The white line of D1 goes to the junction of C1 and R41.

You can now place this in a box of your choice and connect a multimeter and RF connector of your choice. Use the following equation to work out the output power showing on the meter. For best accuracy, size the resistor to match your DC Voltmeter's input impedance: see table below.

$$PWR = \frac{(V_{(read)} + 0.25)^2}{R_{(load)}}$$

R = 4.7 Meg for Zin = 11-Meg; (Supplied)
R = 4.3 Meg for Zin = 10-Meg;
R = 430 K for Zin = 1-Meg;

<u>List of Materials</u>

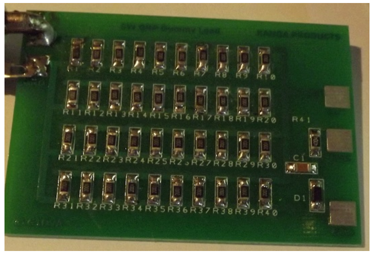
R1 – R40	**2K (size 1206) + 5 Spare**
R41	**4.7M (size 1206)**
C1	**10nF (size 1206)**
D1	**1N5711 Schottky Diode (size 1206)**
PCB	

KANGA PRODUCTS

Kanga “SMD” Dummy Load

Previously kitted and retired by the Four State QRP Group, Kanga Products (UK) have been given permission to re-launch this excellent example of a Self-Training kit in SMD Soldering.
For those who wish to see the original documentation especially those new to SMD Construction, these will remain on the 4SQRP Website http://www.wa0itp.com/dummyload.html.

The Kanga version is a total SMD component kit and is designed to allow the builder to gain experience in SMD soldering.



A dummy load is a resistor equal to the output impedance of a transmitter. It is used to convert the output power to heat instead of feeding it to the antenna.

This has the advantage that transmitter testing and tuning can be carried out without radiating a signal which can cause interference to other band users. Also with the addition of a simple diode detector circuit good accurate power output measurements can be made.

Normally transmitter dummy loads use special low inductance resistors designed to have constant impedance over the operating range. These are expensive and sometime difficult to obtain.

When using Low Power (QRP levels), suitable loads can be made from Low Wattage carbon Resistors.
This is designed so that the load runs cool and not over heating the components. D1, C1 form a half wave rectifier which when used with a high impedance voltmeter determines the RMS reading, but this depends on your DDM (See back page for calculations).

Specifications and Design Features

This is a QRP power level SMD Dummy Load, 40 x 2K ohm 1/4 watt 1206 size SMD resistors are soldered in parallel to make a 50 ohm load. An RMS RF probe is included to estimate output power with your DMM. An oscilloscope for monitoring waveforms can be connected also. The original design survived a 2 hour test at the 4 watt power level.

Construction

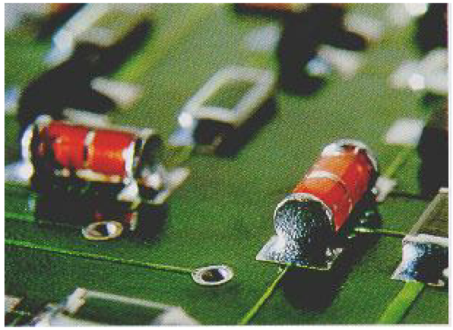
Firstly check to see that you have all the components listed, on checking the 2K ohm resistors you will find five (5) extra, this is case you lose one when soldering to the board.

On the 4SQRP website, there are various methods of soldering SMD components. There is also is a link to Steve Weber KD1JV You-Tube video on SMD Soldering. Here we are going to concentrate on the hand soldering method.

The following describes the hand soldering method of reliably soldering surface mount components with the type of tools previously mentioned in these instructions i.e. fine-tip tweezers, a rosin based flux (a pen is ideal but a cotton bud with liquid flux is just as good), rosin core solder (.04 mm dia. if possible) soldering iron with a small tip. A reel of solder wick if removing components.

Apply a small bead of solder to one of the PCB pads for R1 the component to be installed. Apply some liquid "*rosin based"* flux from a flux pen or other applicator to the solder bead and to the other pad(s) for the component. Next, using a pair of tweezers, pick up the component to be installed and place it over the appropriate pads.

With the part in position, move the soldering iron to the solder bead on the PCB pad. Apply a small amount of heat from the iron to flow the bead, Simultaneously lowering the part against the board and correcting for any rotational misalignment. Remove the iron and allow the solder to cool. Inspect the joint -- at this point you are not concerned about the quality of the actual solder joint, just the positioning of the component. The part should be flush against the PCB, with both ends properly contacting the pads. The part should be straight, and centred between the two pads. Apply a liberal amount of liquid flux (again, a flux pen or cotton bud wetted with Flux) to both ends of the component.



Heat the currently unsoldered end of the component and the adjacent pad with the soldering iron, and carefully wipe on a small amount of solder. You want a small fillet as shown in the illustration, not a large glob of solder.
Make sure that the solder has flowed onto the pad as well as the component lead - don't be fooled by a lump of solder on the end of the component that doesn't flow under the component and onto the PCB pad.