

# Filtered S-Band Driver Amplifier

## Assembly guide

This kit can be assembled in a couple of hours by constructors with a medium level of experience. In addition to the kit you will need a 2.5mm drill for the mounting holes in the heatsink. You will also need a pair of SMD tweezers, a soldering iron with a fine tip and some small diameter solder e.g. 0.4 or 0.5mm. Finally, a headband magnifier or similar will help considerably with the SMD parts. While it is possible to build the circuit in almost any order, this guide provides a proven sequence of assembly. To avoid mis-identifying components, the kit is divided into 2 separate bags. The contents of the bags are given on the packing list. As some components are visually very similar and are not marked, please resist any temptation to simply empty the bags and hope to find the correct component from a big pile of bits. Instead, keep the contents of the bags separate and refer to the packing list as you progress your way through the assembly process. As with all microwave circuits, try not to use too much solder. Any excess can be easily removed with desolder braid.

### Step 1. Mounting the **PCB** onto the **heatsink**.

Start by positioning the PCB onto the heatsink, carefully lining up the edges of the board with the edges of the heatsink. Note the PCB is not square, there are a couple of millimetres difference between edges. The mounting holes in the PCB have been carefully positioned to avoid drilling into the heatsink fins. When you have everything aligned, use a pencil to mark the position of one of the holes. The mounting holes in the PCB are 3.1mm dia and the drill required is 2.5mm. This allows you to use either the stainless steel self tapping screws provided, or if you prefer, 2.5mm is the correct size for a 3mm tap. Drill the first hole, secure the PCB into position and check the alignment with the edges of the heatsink. When you are happy with positioning, mark then drill the remaining 3 holes. Finally, de-burr or lightly sand around the holes in the heatsink and then refit the PCB, tightening the four screws one at a time. If all is well, it's time to start soldering components.

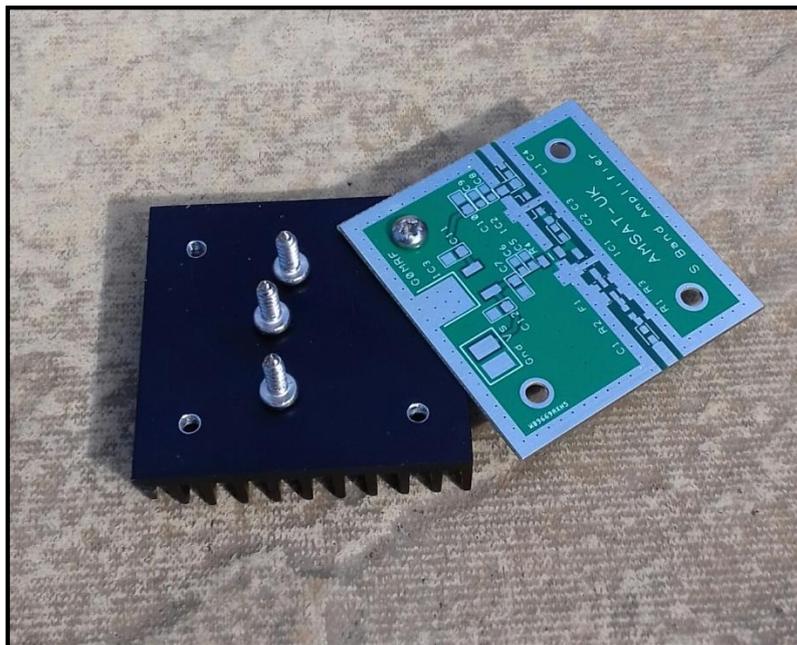
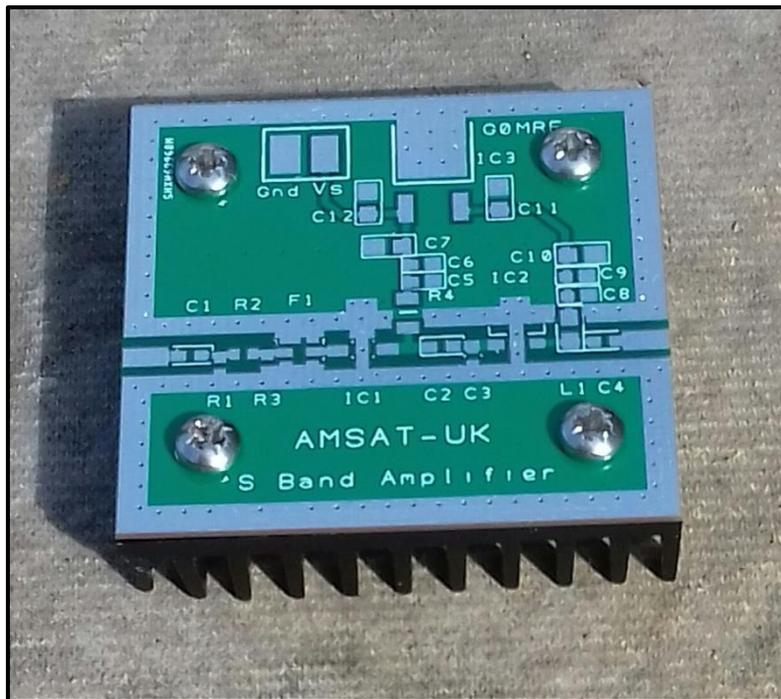


Figure 1 Drilling the heatsink



**Figure 2 PCB correctly mounted onto the heatsink**

**Step 2. IC3 C11 C12 C6 and C9**

From Bag 1, remove 78M05 voltage regulator IC3 and the 2 x 2.2uF 0805 size capacitors. Starting the assembly with some of the larger components is good for confidence and it allows you to get use to soldering SMD before moving onto some of the more challenging components. Locate the correct position and solder IC3. The ground connection is the tab on the component, you may find you need a little higher temperature to heat the tab and the PCB ground before the solder flows correctly. Next remove the capacitors from the packing by gently lifting the thin plastic covering on one side. Don't be tempted to grip the capacitors too hard. If they slip out of the tweezers they can be launched a considerable distance from the workbench and never be seen again. Solder them into the positions adjacent to the voltage regulator. Next remove 1nF 0603 capacitors C6 and C9 from their packaging. These are the smallest size used, but can be easily soldered albeit with a little practice. I usually apply a very small amount of solder to one of the pads, then hold the capacitor over or next to the pad. Then heating the solder, I slide the capacitor into place. If all is OK and the capacitor is flat against the PCB, then I solder the other end. Importantly...I then go back to the first joint again, adding a little more solder. Hopefully, both ends are now soldered correctly.

**Step 3. C1 C4 C5 C8 and C2**

Remove the strip of 4 x 22pF capacitors from bag 1. Solder supply decoupling capacitors C5 and C8, then input and output coupling capacitors C1 and C4 into position. Finally locate C2 in bag 1 and fit it onto the board. It's worth mentioning that this capacitor is the only 'critical' component. It performs input impedance matching to the final amplifier and has a tolerance of +/- 0.1pF. Pads are provided on the PCB for C3, a low value capacitor. However, testing revealed that no component was necessary. Perhaps the stray capacitance from the FR4 board was sufficient.

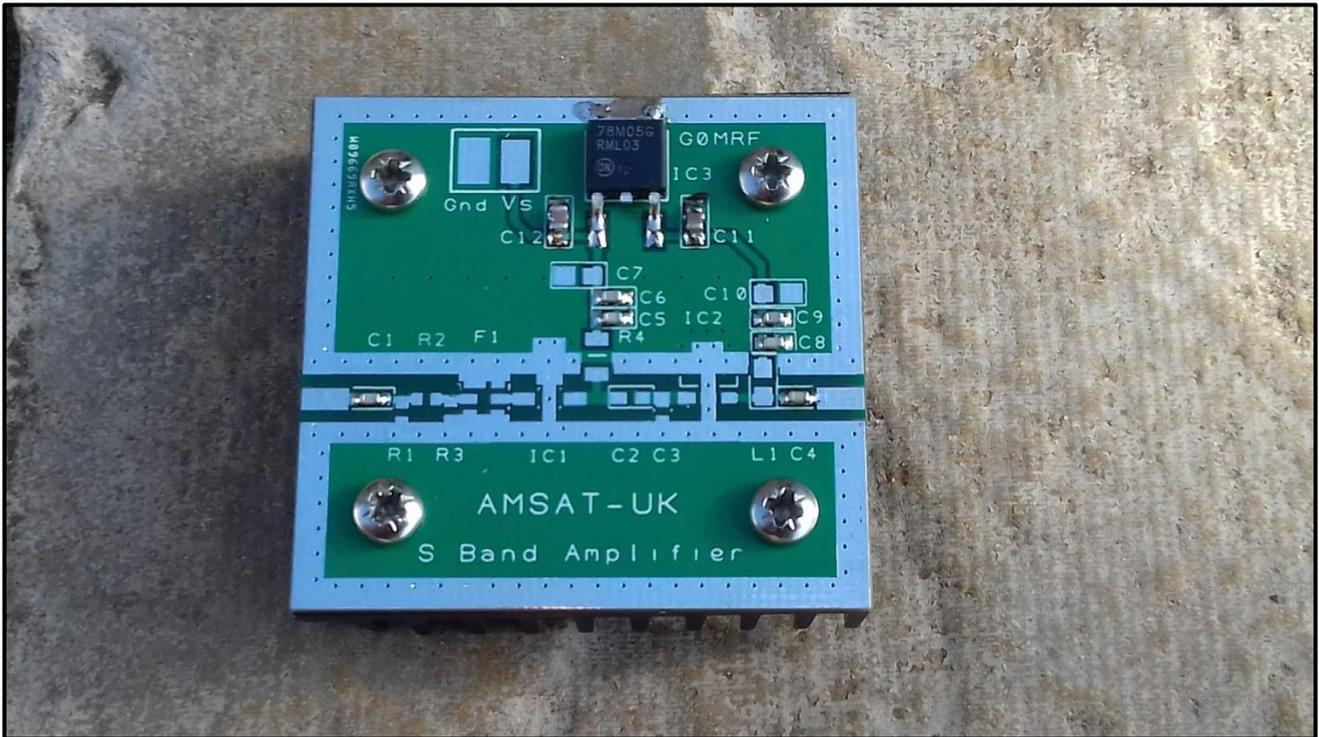
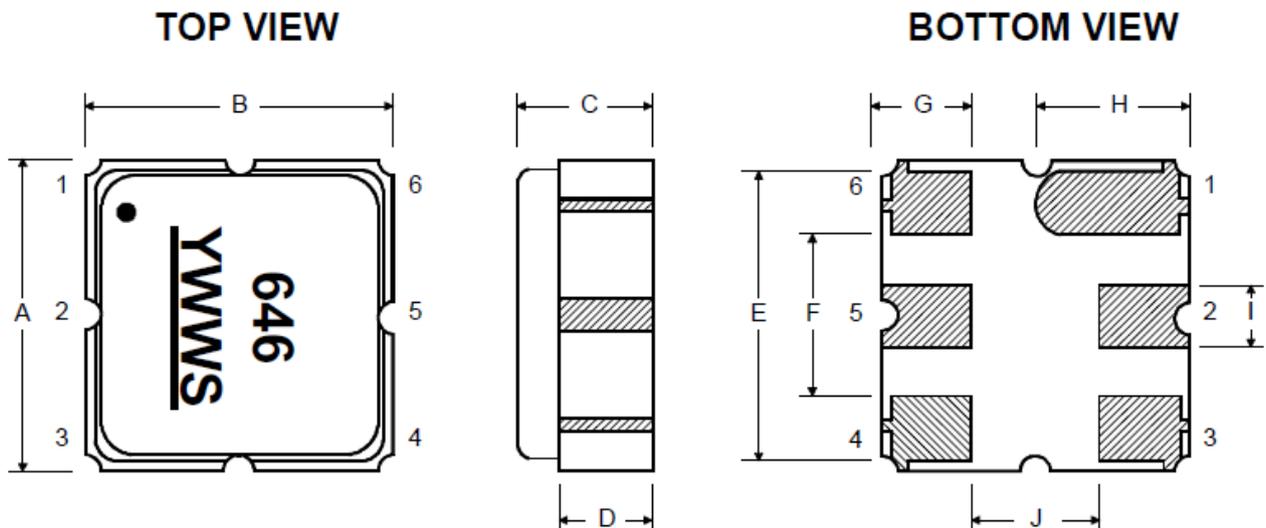


Figure 3. PCB with Voltage Regulator, 2.2uF 1nF and 22pF capacitors installed.

Step 4. **Bandpass Filter F1 ( 2400 – 2500MHz )**

The bandpass filter needs to be installed before R2 and IC1 and requires some precise soldering. The SF2124E has 6 connections; 4 ground, an input (pin 2) and output ( pin 5) Pin 1 is marked with a dot on the top of the package. In the photo above, pin 1 is the top left pad.



My method, which may not be the best but works for me, is to apply a small amount of solder to the pad on the PCB for the output connection. If the pad is heated and solder applied quickly, then the soldering iron can be removed while there is still some flux on the board. The filter can be positioned correctly and held. It will be raised off the PCB slightly because of the solder under pin 5, but if the pad is reheated, the solder melts and the filter drops down slightly into the correct

position with pin 5 soldered to the board. Now look very carefully at the position of F1. Is pin 5 in the middle of its pad? Look at pin 2. Is it in the correct position above its pad? If you need to move F1, now is the time. When you are happy that the filter is in the correct position, solder pin 2, then make a better job of pin 5. If all is OK, solder the ground connections by heating the groundplane next to the pin and apply solder letting it flow under and onto the edge of the filter. With the filter soldered in place you are now on the final part of assembly which will be relatively easy !

#### Step 5. R2 L1 C7 C10 + R4

These components are in bag 2. Remove from their packaging one at a time and fit to the PCB. R2 in the kit is 0 Ohms. If you have a higher drive level, then a 50 ohm attenuator can be fitted at R1 R2 and R3. The value of R4, is 158 Ohms and this is achieved by using 2 x 316 Ohm resistors in parallel. This is necessary for power dissipation. Solder one 1206 sized resistor onto the pads for R4 then place the second resistor on top and solder in place. Finally solder C7 and C10 into position.

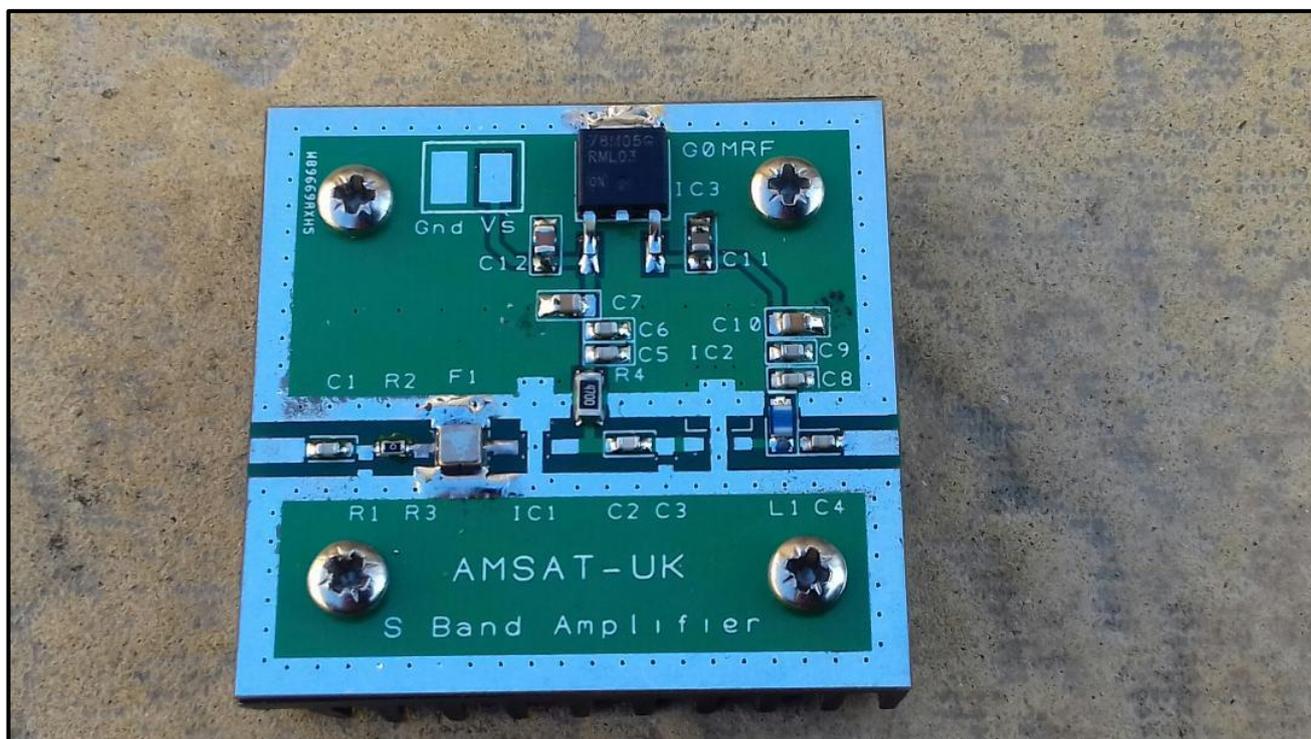


Figure 4 Added Bandpass filter, R2 R4 L1 C7 + C10

#### Step 6. IC1 and IC2

The final components are the two integrated circuits IC1 and IC2. They are both the same SMD package but the Gali5+ can be identified by the number 5 printed on it. It's important to ensure there are good ground connections under these devices. I soldered the output pin, then the input and finally, using a larger soldering iron heated the groundplane next to the tab and allowed solder to flow under the device. This worked, but I'm not sure it's the best method as you are never quite sure if the solder has gone under the tab as well as you hoped. Solder paste would be better, or perhaps cut a short length of solder, position it under the tab. Hold the device in place then heat until the solder melts and the IC drops that last 0.5mm into position.

### Step 7. SMA sockets

The SMA sockets supplied have a PTFE sleeve covering a portion of the centre pin. This needs to be removed using a Stanley or modelling knife. Trim it back until it is flush with the body of the socket. Reduce the length of the centre pin with a pair of cutters so that it is about 3-4mm long. When held in position against the edge of the board there should be a gap between the end of the pin and the solder on capacitors C1 at the input and C4 at the output.

Holding the SMA socket so the body is against the PCB and the pin in in the middle of the track, solder the pin and then look carefully at your work. Is the body horizontal or does it need rotating slightly? Is the body as close to the PCB as possible? If all is correct, you can now solder the body of the SMA to the PCB by running a fillet of solder along the join on both sides. – Well done, you have completed the assembly of your driver amplifier.

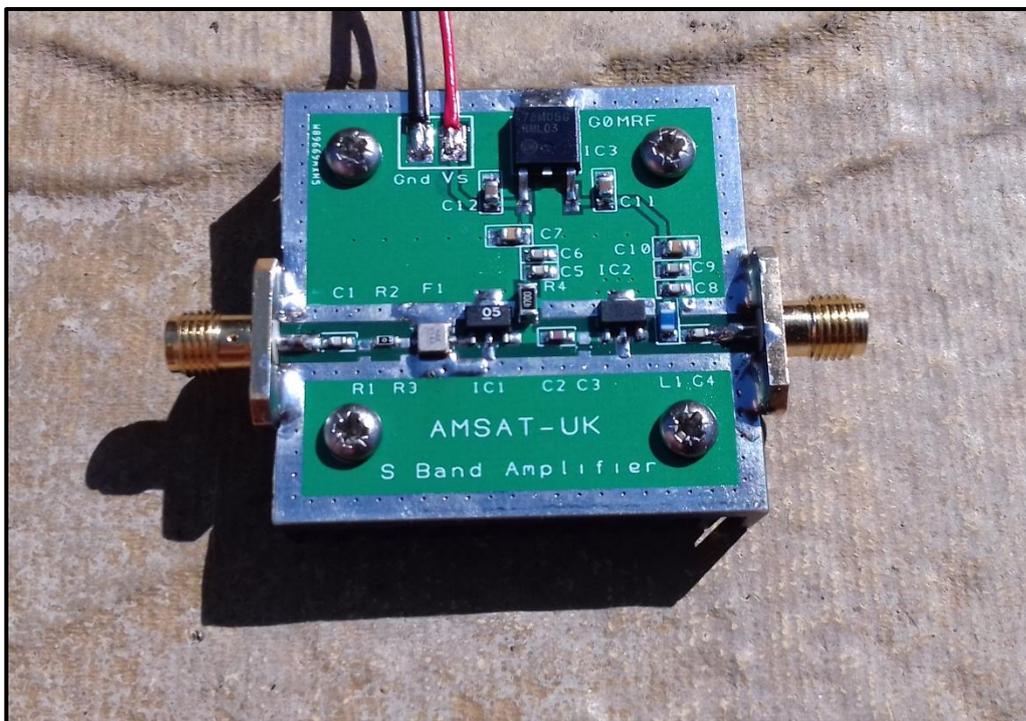


Figure 5. Completed driver amplifier

## Testing.

Solder wires to the pads for Vs and Gnd. If you have a 50 ohm load, attach it to the output, if not, then initial DC testing without a load will probably be OK. Having built a number of these boards I've not seen any sign of instability. Place an ammeter in series with the positive supply and without switching on, adjust the power supply to 12V. If you have a current limit, set it to about 250mA. Check the power leads are not reversed...and switch on. Hopefully you will see about 150mA. This is a good indication that both amplifiers are operating correctly. I would suggest running the board from 12 or 12.6V. At 13.8V the supply current and power dissipation increase without any significant improvement in performance. If the DC test is fine, you can now check the RF performance. Remember this amplifier is designed for the Satellite sub-band, it will not work on 2300 or 2320MHz terrestrial allocations.

The completed board will have a gain of about 25 - 26dB and for best linearity, should be kept to 200mW Output. If you have access to suitable test equipment, you will be able to measure the performance of your driver amplifier.

If you don't have any test equipment, then looking at supply current will provide some indication of correct operation. With a load attached to the output and a 2400MHz drive source, use a multimeter to look at supply current. Start at a low drive level and gradually increase the drive. The supply current remains constant up to about 125mW RF output. As power output increases beyond this level, the current to the Qorvo TQP7M9101 will increase. At 200mW output and higher, you will see the total current increase by about 20mA. This is normal behaviour for the device and does not indicate non linearity or over driving, but don't go too far. As always, pushing a device to its limits will cause non linearity and or damage. If in doubt, you can always take a look at your transmitted signal via the QO-100 Goonhilly Web SDR.

<https://eshail.batc.org.uk/nb/>

Qorvo TQP7M9101 400 - 4200 MHz, 0.25 Watt High Linearity Amplifier

<https://www.qorvo.com/products/p/TQP7M9101>