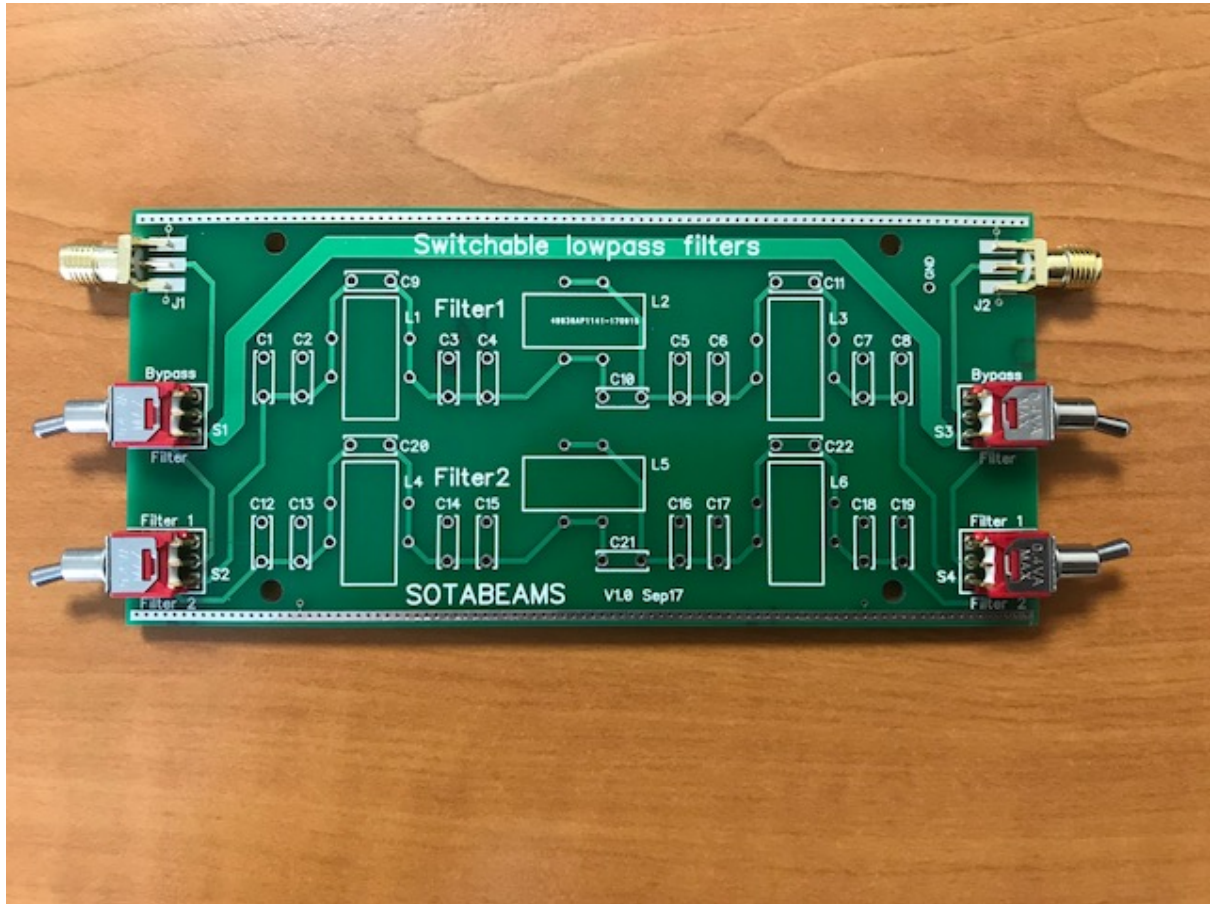


The two band filter PCB has been designed to allow users to implement their own filter designs. Assembly is simple - see below.



### Information for self-designers

The filter board has been designed to accommodate a standard 7 element low-pass filter (suggested maximum frequency 30 MHz). The design size for the toroids is T50 but T68 will also fit (not in enclosure). The capacitor footprint is for a 5 mm lead spacing. For good performance (low loss and good attenuation) COG or NPO dielectric capacitors should be used (silver mica or polystyrene are also good). For flexibility there are footprints for two capacitors in parallel and there are also footprints for capacitors to be placed in parallel with the inductors to allow more complex filter designs to be implemented.

Plated-through holes with cleared roller-tinned copper have been placed along the sides of board. There are also five cleared pads on the ground-plane that can be used for sprung earth contacts if required.

Dimensions 120 x 55 mm. Board includes 4 x 3 mm mounting holes.

The enclosure is equally simple to assemble. Carefully remove the protective paper from the end plates before use. The four rubber fit should be on the plain side of the enclosure. In use, the switches need have the same selection at both ends of the filter. You could mark which bands the filters cover on the underside of the unit.

We sell a range of filter component kits that can be used with this PCB.

<https://www.sotabeams.co.uk/band-kits-for-low-pass-filters/>