**Dramatically improve performance of SDR – especially at HF**

# TUNABLE HF PREAMPLIFIER WITH GAIN CONTROL

**by Charles Kosina**


**There are many cheap Software Defined Radio (SDR) modules available which perform brilliantly at VHF/UHF but they generally have poor HF (3-30MHz) performance. They also suffer from wide-open front ends, which makes them susceptible to cross-modulation from strong signal sources. This simple tunable preamplifier greatly improves SDR HF performance. It has (optional) gain control and can run off a 5V supply or phantom power.**

**Most SDRs (and many other radio receivers)** can benefit from a preamp to boost the signal from the antenna.

This one is nice and simple, low in cost, easy to build and works well over most of the HF range.

It can be built with variable or fixed gain. Variable gain is ideal as it allows you to avoid overload on strong signals, while still taking advantage of the improved selectivity of a tuned front end.

It's a fairly compact unit when completed, and runs from a 5V power supply, which in some cases can come from the receiver itself via the Preamp's output lead, using 'phantom power'.

The circuit of the HF Preamp is shown in Fig.1.

The input signal is fed into chassis-mount BNC connector CON1, then to the PCB via pin header CON2 and onto DPDT switch S1, which passes it to one of two transformers. This provides two different tuning ranges, allowing the tuning to be more selective.

T1 covers a range of about 5-11MHz, while T2 covers 11-24MHz. Both are tuned by dual variable capacitor VC1, with its two gangs wired in parallel to give a 6-200pF range.

The tuned signal is then fed to gate 1 of dual-gate Mosfet Q1. The signal is DC-biased from the nominally +5V rail via a 150kΩ resistor and 10nF low-pass filter capacitor, to reject supply noise.

The Mosfet's gain is controlled by varying the DC voltage on the second gate, using potentiometer VR1 which has padder resistors at either end, to limit its wiper voltage to the useful range.

Fixed gain can be provided by omitting VR1 and changing the resistor values, as described in the circuit diagram.

Q1's drain load is the primary of transformer T3, with a 1.25mH inductance. The other end connects to the +5V rail which is bypassed by a 10nF capacitor. The 75µH secondary is connected similarly, and the signal is AC-coupled to output SMA connector CON3 via another 10nF capacitor.

Alternatively, if the device is to be phantom powered via CON3, jumper JP1 is inserted, allowing the DC sup-

### *Features & specifications*

*Tuning range:...... 5-24MHz in two ranges (wider tuning range possible)*
*Bandwidth:.......... typically 50-250kHz (varies with tuned frequency)*
*Gain: .................. typically 34-36dB*
*Power supply:..... 5V DC @ 30mA*
*Connectors:......... BNC input, SMA output (can be varied)*

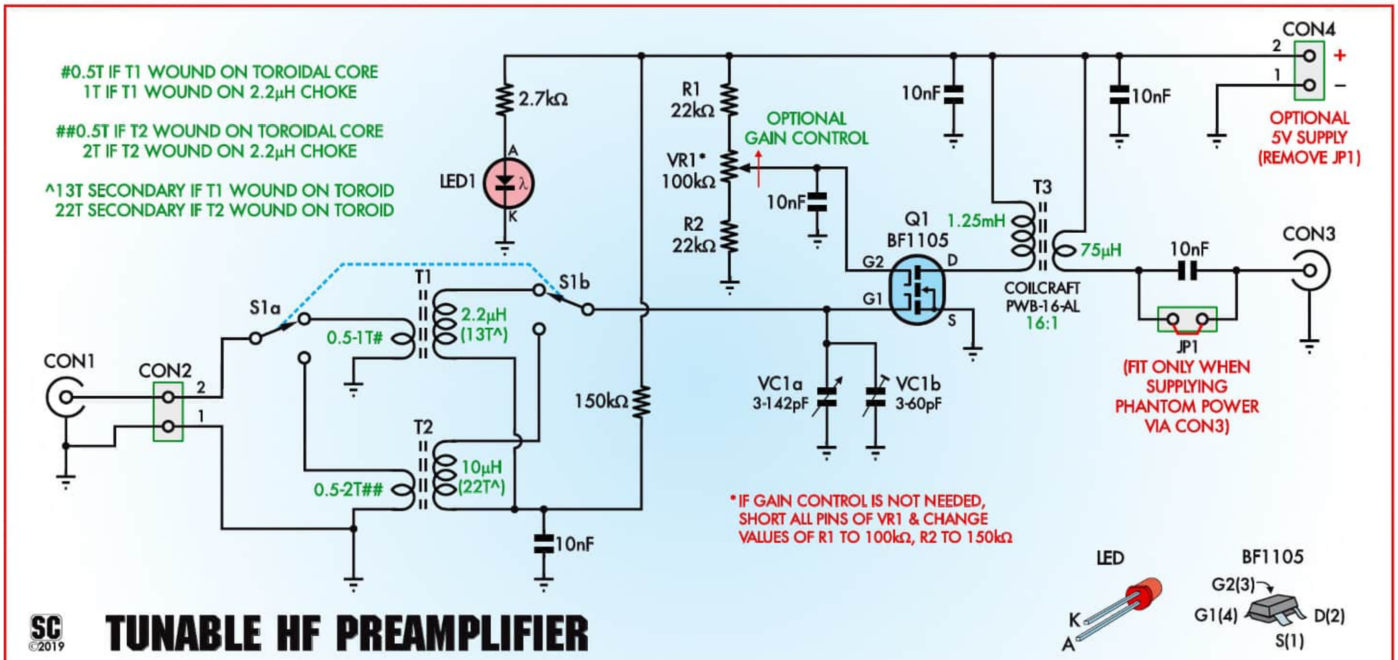



**Fig.1: the circuit is quite simple, especially given its performance. It has a gain of around 35dB and a tuning range up to about 24MHz as shown (but can be extended to about 30MHz). VC1 a and b is a miniature dual variable capacitor, typically sold as a tuning capacitor for small radio receivers.**

ply voltage to flow through T3's secondary and into the +5V rail. This is then modulated with the output signal which is coupled in from T3's primary.

## Two versions

You can build the device in two different versions. Version 1 has T1 & T2 wound on toroidal ferrite cores. These are not that easy to get, and winding the turns it tedious, but they have the advantage of a very high unloaded Q, up to 350.

Version 2 is easier to build as T1 & T2 are based on readily-obtainable axial RF inductors, which are each about the size of a 1W resistor.

The primary winding is just one or two turns of wire around the inductor body. These inductors exhibit a surprisingly high Q, up to 120 in the range of interest.

## Obtaining the parts

The output transformer is a broadband Coilcraft device. I got mine as a free sample, but they are also readily available from element14. The tuning capacitor comes from Jaycar and many other sources, including eBay. The SMA output connector is readily available on eBay, with one local seller listing ten for $6.59.

The other components are reasonably standard parts. Those which are not available from Jaycar or Altronics can be purchased from Digi-key, Mouser, element14 etc.

## Changing the frequency range

If you changed the 2.2μH inductor to 1μH, that would give you a tuning range of about 12-30MHz, giving you almost full coverage of the HF band. If building Version 1, with the toroidal ferrite cores, this could be achieved by reducing the number of secondary windings on T1 by about one third. If building version 2, using RF chokes, simply substitute a 1μH choke.

## Construction

The Tunable HF preamp is built on a double-sided PCB coded CSE190502, measuring 79.5 x 29mm. Refer to the overlay diagram, Fig.2, along with the photos to see how it all goes together.

Fig.2(a) shows Version 1, with T1 & T2 wound on ferrite toroidal cores, while Fig.2(b) shows Version 2, using the RF chokes with turns of wire around the outside of each to make them into transformers.

We used 0.25mm insulated wire but enamelled copper (ENCU) wire would also be satisfactory.

Many of the components are SMDs, with 2012 (metric) / 0805 (imperial) capacitors and 3216 (metric) / 1206 (imperial) resistors.

I find that an SMD board now takes me less time to assemble than one with through-hole components, and none of the parts on this board are difficult to solder.

Start by fitting the SMD passives. Tack one end down, then solder the other end and wait for the joint to solidify before refreshing the first joint. Then mount dual-gate Mosfet Q1 with its larger tab orientated as shown above, followed by transformer T3, with its pin 1 dot at upper left.

Follow with edge-mount connector CON3, which is placed over the edge of the board before soldering its pins top and bottom. Make sure the middle contact pin is on the correct side to match with its pad. Then fit the pin headers where shown.

If you are building Version 1, now is the time to wind and mount the toroidal transformers. T1 has a half-turn for its primary (best fitted after the secondary has already been soldered to the board) and 13 equally-spaced turns for its secondary. Try to wind the secondary so that it spans just over half the core, meaning the start and end correspond with the PCB pads (see photos).

T2 also has a half-turn primary but a 22-turn secondary, which is wound to cover the entire circumference of the core (not shown for clarity in Fig.2(a); see the photo) and then brought back across the core to terminate to the other secondary pad on the PCB.

Once you've wound the secondaries

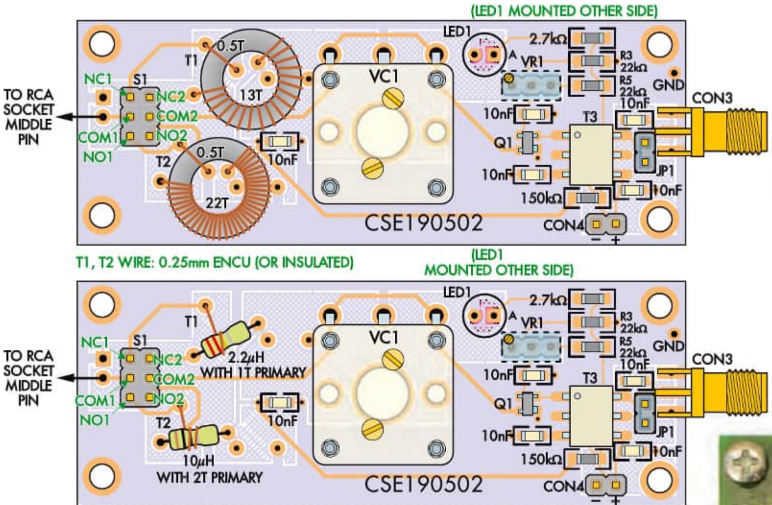
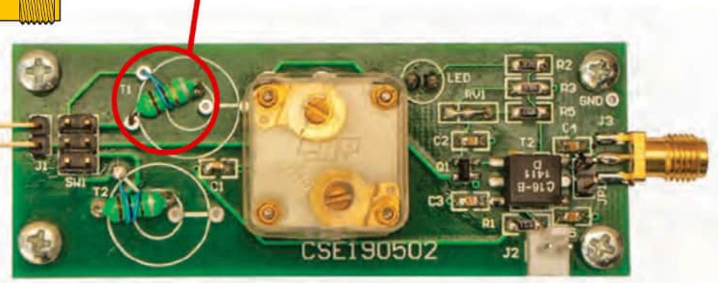



Fig.2a (top) is the component overlay for version 1, using two toroids for T1 and T2 with primaries and secondaries wound through them. Fig.2b (bottom) shows version 2, an identical overlay but using axial RF chokes instead, with primaries of one or two turns of thin wire around them.


**The same-size photo below shows version 2, with the enlarged inset at left showing how the one and two-turn primary windings are added. The PCB pads for the "earthy" end of the primaries are directly under the 2.2μH and 10μH chokes.**

and soldered them to the PCB pads, you can solder one end of each primary, pull it tight across the core and then trim it and solder the other end.

If you're building Version 2, you just need to wind one turn of 0.25mm wire (ENCU or insulated) around the body of the 2.2μH inductor and fit it for T1 as shown, with the added windings as the primary, and wind two turns around the 10μH inductor and use it as T2; again, the added windings are the primary.

If you're using a trimpot for VR1, fit it now. If you want the gain to be externally adjustable, solder leads onto the three terminals of your chosen potentiometer and attach a three-pin plug to the other end. Alternatively (and more simply), cut female-female jumper leads in half and solder the exposed ends to the pot terminals. The sockets at the other end can be plugged into the PCB header later.

Now fit the variable capacitor. Remove the knob first, then attach the body to the PCB using the two supplied screws through from the underside. Solder the three pins, then re-attach the knob to the shaft, which passes through a hole in the PCB.

Leave LED1 off for now.

## Preparing the case

Now place the PCB assembly in the case, sitting on its spacers, and slide it so that CON3 touches the side of the case. Measure the distance from the centre of CON3 to the top of the box. Then measure that same distance on the outside, from the top of the box near CON3, and mark where the hole will need to be drilled. Remove the PCB and drill a small hole there, then enlarge it to 7mm.

Check that the connector fits through the hole with the spacers sitting on the bottom of the box. If so, deburr it. Otherwise, you may have to enlarge it slightly. Once it fits, drill a small hole at the opposite end of the box and enlarge it to around 10mm, then check that the BNC socket fits. Once it does, deburr that hole too and again, clean out the swarf.

Now remove the spacers from the PCB, push CON3 through the hole you drilled and mark out the four mounting hole positions. Also mark the location where LED1 will protrude through the base, once it has been installed, and mark a suitable location for the DPDT switch. Note that a 5mm LED will have to clear the PCB once fitted.

Drill the marked holes to 3mm, then enlarge the LED hole to 5mm, and the switch hole until the switch fits. Deburr all the holes and clean off the swarf. If you're building the Preamp with an external gain control, now is also a good time to figure out where the pot will be mounted and drill and deburr a suitable hole.

If you are going to be supplying external power, drill a hole for the DC socket now. It would make sense to move the BNC socket slightly towards one side of the case to make more room for the DC socket.

## Final assembly

The last component to be fitted to the board is the LED. It's mounted on the opposite side to most of the other components, and its longer lead must face towards the pad marked "A" on the PCB.

Push its leads through their holes so that the lens is fully down onto the PCB, then slot the board in place holding the leads, and use them to push the LED lens through its mounting hole while CON3 is hard against the edge of the case.

Prop the board up so that the LED lens is not being pushed back into the hole, attach a couple of the board mounting screws to ensure it's in position, then solder and trim LED1's leads. After that, insert the remainder of the PCB mounting screws.

Mount the BNC socket in the hole you made earlier and solder a short length of hookup wire to its middle pin. Connect this wire to the lower terminal of CON2, to the left of the header for S1, as shown in Figs.2(a) & (b). You don't need to connect the RCA socket shield, as it's grounded to the metal box and this connects to board ground via CON3's shell.

All that's left now is to wire up and fit switch S1. Crimp a length of 6-way ribbon cable into the IDC connector shell, so that the red wire will be towards the top when plugged into the header on the board such that the cable exits to the left (ie, towards the nearest board edge).

Now separate and strip the wires at the other end. Starting with the red wire, solder them to the following switch terminals: NC1, NC2, COM1, COM2, NO1, NO2. In this case, the numbers 1 and 2 refer to the two switch poles. It doesn't matter which is 1 and which is 2, as long as you are consistent. It also doesn't matter which side of the switch you consider to be NC and which is NO.

Once the wires have been soldered and the switch mounted in the base, plug the IDC socket into the header as shown in the photos.

If using a DC socket to feed in external power, solder wires to its two tabs; if your socket has three tabs, plug in a plugpack and use a DMM to figure out which is positive and which is negative. Mount the socket in the hole you made earlier, then terminate the leads to CON4, either by soldering them directly to its pins (see PCB for polarity) or by attaching a two-way header socket to the wires.

As with the pot, you can cut a female-female jumper lead in half and then solder its bare ends to the DC socket. The other ends will plug straight into CON4. Alternatively, if using phantom power from the radio receiver via CON3, place a jumper shunt on JP1 now.

If you're fitting an external gain control pot, mount this now, and plug its terminals into the pin header soldered in place of VR1. The lead soldered to the anti-clockwise end of the pot (as viewed from the front) plugs into the left-most terminal of the VR1 header, with the PCB viewed right-side-up.

## Parts list – Tunable HF Preamp

1 double-sided PCB, code CSE190502, 79.5 x 29mm
1 diecast aluminium case, 115 x 65 x 30mm [Jaycar HB5036, Altronics H0421]
1 BF1105 dual-gate SMD Mosfet (Q1)
1 5mm or 3mm LED (LED1)
2 small toroidal ferrite cores, 12.5mm OD, 7.5mm ID, 5mm thick (T1/T2) [eg, TDK B64290A0044X830] ***OR***
2 axial RF chokes, 2.2µH & 10µH [Jaycar LF1514 + LF1522, Altronics L7014 + L7022]
1 Coilcraft PWB-16-AL transformer (T3) [element14]
1 chassis-mount BNC socket (CON1)
1 edge-mount SMA socket (CON3)
3 2-pin headers (CON2,CON4,JP1)
1 chassis-mount DC socket (optional)
1 shorting block/jumper shunt (for JP1)
1 DPDT toggle or slide switch (S1)
1 3-pin header (for VR1)
1 3x2-pin header (for S1)
4 6.3mm Nylon M3 tapped spacers
8 M3 x 6mm machine screws
1 1m length of 0.25mm diameter enamelled copper or insulated wire
1 1m length of light-duty hookup wire
1 50mm length of 6-way ribbon cable (for S1)
1 6-pin IDC socket (for S1)

**Capacitors**
5 10nF 50V SMD ceramic capacitors, 2012/0805 size, X7R dielectric
1 dual variable capacitor (VC1) [Jaycar RV5728]

**Resistors** (all SMD 3216/1206 size, 1%)
1 150kΩ    2 22kΩ*    1 2.7kΩ
1 100kΩ linear chassis-mount potentiometer (VR1) ***OR***
1 100kΩ multi-turn vertical trimpot (VR1)

* or 1 100kΩ + 1 150kΩ for fixed gain (omit VR1 & 3-pin header)

### Using it

Now it's just a matter of screwing the lid onto the box, connecting your antenna to CON1, your radio to CON3, hooking up a 5V power supply (if using external power), and switching S1 to the appropriate band. You may wish to label the case to indicate which position is for the lower tuning range and which is for the upper.

With power applied, check that LED1 lights. Switch to your SDR's spectrum analyser view and set the range to 3-30MHz. Check that adjusting VC1 changes which frequencies are being amplified, and that VR1 (if fitted) allows you to control the gain. Check also that S1 switches bands and that the two ranges are roughly as expected.

As VC1 is not calibrated, you will need to use a spectrum display to see what frequency you are tuning in, although you can 'blind tune' by simply adjusting VC1 and S1 for maximum signal at your desired frequency.

Then adjust VR1 (if fitted) for the best reception without overloading the receiver. **SC**


**Shown a little larger than life size, this is the completed PCB (in this case version 1 with toroids) mounted in the diecast case. S1 is shown here mounted off the board but the Altronics S2075 slide switch could probably be mounted directly.**