The QRP2004 all-band HF Transceiver

Designed by the QRP2004 Design Team
Alan Rowe M0PUB, Jan Verduyn G0BBL & John Law G8BTR
Copyright © 2004 - 2005 by QRP2004 Design Team
QRP2004 Web Site: http://myweb.tiscali.co.uk/qrp2004/

Introduction

The QRP2004 is an all-band Direct Conversion Transceiver aimed primarily at amateur use on the HF bands. It supports CW, LSB and USB modes on all HF amateur bands from 160m to 10m. It also offers general coverage receive capability, with continuous tuning from 132kHz to 30MHz with a 10Hz tuning resolution. As presented here, the design also offers optional support for the 6m band, but please note that this is still at an experimental stage, and at the moment 6m performance is relatively poor.

Probably the single feature which sets the QRP2004 apart from other designs is its received audio quality, which has been widely praised by those that have heard it. However, it provides an all-round high standard of receive performance including excellent selectivity and good sensitivity. Although we have not been able to measure IP3 figures yet, the mixer and front-end design should ensure good performance in that respect too (we anticipate somewhere in the region of +20dBm, based on results from similar mixers).

Output power is 5W on all bands apart from 6m – hence the "QRP" moniker. Of course, it can be used with an external PA if you require more power.

The QRP2004 is not a simple project, is not recommended for beginners, and is not available as a kit. However, we have provided full schematics and tried-and-tested PCB layouts (on the internet) to enable a reasonably proficient home-brewer to reproduce the design. Work still continues to improve some aspects of the radio, and the current state of the design is published here to encourage others to experiment with this approach to SSB reception and transmission.

If you are looking for a high performance radio that comes with step-by-step construction notes then we recommend that you look no further than the excellent Elecraft K2. However, if you are interested in more of a challenge and want to build something which is distinctly different from other radio designs, then you might find that the QRP2004 fits the bill.

Performance summary

Receive sensitivity (for 10dB SINAD)	0.2 – 0.35uV, depending on band
AGC range	70dB
SSB bandwidth	Variable, 1.1kHz – 2.2kHz
CW bandwidth	Variable, 400Hz – 1kHz
Rejection of unwanted sideband	45 – 38dB, depending on band
Transmit power	5W

Please note that these figures only relate to HF performance. Performance on 6m is currently much reduced, with about 1uV input sensitivity, 1.5W output power and 15 - 20dB opposite sideband suppression.

A brief history

The QRP2004 design has a very long history, starting back in 1998 with the QRP2000 which was developed by G0BBL and Steve Farthing, G0XAR. The QRP2000 was based on the excellent R2/T2 designs from KK7B, and offered a high level of performance...but it suffered from a number of disadvantages. The main problem was that it required two AD9850 synthesizer chips, and that was making construction very expensive (especially now that the AD9850 is on allocation and the UK price for small quantities has doubled to around £35 each). The design also lacked a number of features which we considered useful - most notably AGC circuitry.

The original design for the QRP2000 can still be found on the internet (Ref 1). Despite its limitations, it resulted in a presentation at the FDIM QRP convention at Dayton in May 1999 (Ref. 1a), and some very useful feedback from other constructors.

Immediately after the 1999 Dayton convention, we began work on the QRP2001. The first prototype of the new design was demonstrated at Rochdale QRP Convention in November 1999. Technical details were also published in Sprat magazine, Winter 1999/2000 edition (Ref. 2) and presented at the FDIM QRP convention at Dayton in May 2002. The QRP2001 receiver was pretty popular with those who built or borrowed it, but despite a lot of further development work, it stubbornly refused to work as an acceptable transmitter. Again, the QRP2001 receiver design can be found on the internet (Ref. 3).

The QRP2004 is a major reworking and upgrade of the QRP2001. In addition to finally achieving reasonable transmit performance, many other aspects of the design have been improved. Major new features include variable audio bandwidth under microprocessor control, wider AGC range, and optional support for the 6m band.

Design Overview

Like the QRP2001 before it, the QRP2004 is a direct conversion design which uses phasing techniques to give single-sideband receive and transmit capability. We achieve a high level of performance by utilizing a 'Tayloe' product detector as our one-and-only mixer along with a passive polyphase filter. The excellent strong-signal performance of the Tayloe product detector and carefully selected gain distribution results in a pretty good receive performance by any standards, and a quite remarkable performance for a relatively simple design with wide pre-selection filters.

(Although other designs based on the Tayloe/Polyphase combination have appeared since, we think that the QRP2001 was the first to use this configuration.)

Please note that the Tayloe Product Detector is a patented design, and therefore any commercial use of the design is subject to agreement. The patent application can be downloaded from the US Patent Office web site under patent number 6 230 000 titled "Product Detector and Method Therefore" (Ref. 4). Thanks to 9A2HL for this information.

The output from the polyphase filter passes through a series of audio amplifiers and further filters to define the audio bandwidth. A particularly neat feature of the QRP2004 is the SCAF (switched-capacitor audio filter) which is directly driven by the main control processor. This offers flexible and configurable selection of audio bandwidth at very little additional cost.

If you are familiar with other direct conversion transceivers, then the top level block diagram (Fig. 1) shouldn't be too much of a surprise:

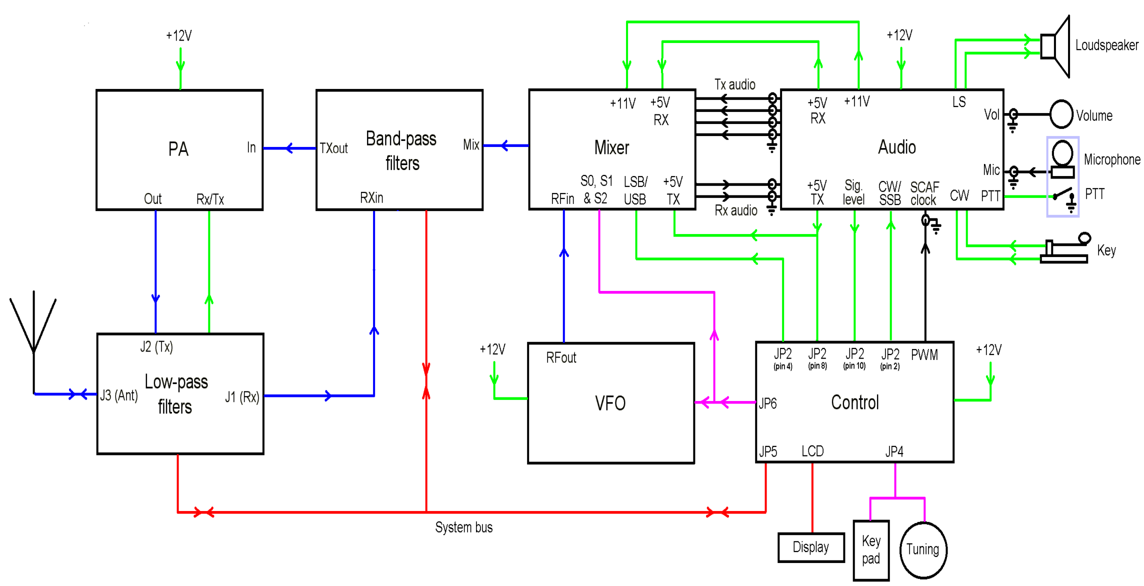

Fig. 1: QRP2004 Block Diagram

The QRP2004 transceiver was designed in a modular fashion, and is intended to be built that way. There are seven separate circuit boards involved, corresponding to the seven major blocks above (actually there are eight PCBs because the VFO is split, but we'll come to that later). The rest of this article describes each of the boards in turn, together with the chassis which we used to create our prototypes.

In fact, this is probably the right point to mention the chassis, which is an important part of the QRP2004 design. Being a direct conversion rig, the QRP2004 provides over 100dB of gain when receiving weak signals, and all of this gain is at audio frequency. We have therefore put a lot of effort into keeping the whole design stable, and the chassis design and board layout are significant factors in this. Early attempts to build the QRP2004 on a simple aluminium chassis resulted in a frustrating tendency to oscillate: try your own chassis designs by all means, but don't say that we didn't warn you!

We have included pictures of our prototype boards where we think these are useful. Where possible we have shown latest revision boards, but this has not always been practical due to publishing deadlines, so you may spot some minor differences between the pictures and the latest PCB layouts.

For ease of testing, and improved chances of success, we recommend that you build the boards in the order in which they are described. We'll start with the control board, but before we do so, here are a few general building tips which apply to all of the boards.

General Construction Notes

All of the QRP2004 PCBs are made from double-sided PCB stock. They are intended for constructors to etch their own, and to help with this we have kept all tracks on one side of the PCB. The other side of the board is unetched, and forms a continuous earth plane. This avoids the need for accurate registration between the upper and lower planes, which can be tricky.

All of the PCB layouts and component overlay diagrams can be found on the internet (Ref. 11).

The boards hold a mixture of leaded and surface-mount (SMD) components. In general we have used leaded components for most things, but SMD for decoupling capacitors, high-speed devices (e.g. the logic chips on the mixer board), or where leaded equivalents are hard to obtain. Leaded components are mounted on the earth-plane side of each PCB, with holes drilled through to the track side. SMD parts mount directly on the track-side of the boards. (The PA board is a minor exception to this – see later).

Of course, with the top copper plane of the board forming an earth-plane, most of the holes through to the track side of the board need to be countersunk on the earth-plane side so that component leads don't make contact with earth. However, some of the holes are specifically provided for wire links between planes, so be careful not to countersink these!

The clue is on the PCB layouts – square pads on the track side indicate links to the earth plane, and circular pads indicate component leads. We recommend that you fit all earth links first, and then countersink the remaining holes on the earth plane side. As an example, Figure 2 shows one of our mixer boards at an early stage of construction.

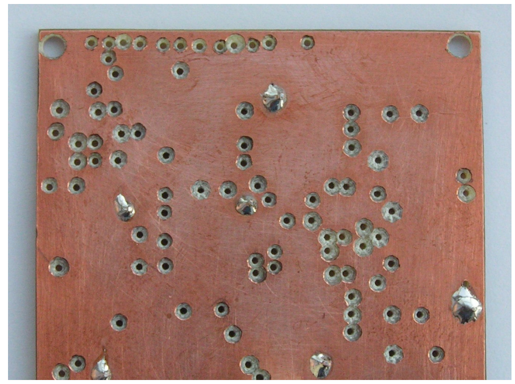

Fig. 2: PCB ready to start adding components

Component leads which need to be connected to earth are usually soldered directly to the earth plane; there is no corresponding hole for the lead. However, in just a few cases component leads are soldered on both sides of the board so that they double-up as earth links. Again, such connections are shown as square pads on the track side. If you encounter one of these and you have already fitted the earth link in place then don't worry: it's probably easiest to leave it and just attach the relevant component lead directly to the earth plane.

Since we only have tracks on one side of our boards, it has been necessary to use a few wire links on most boards. The position of the links can be easily seen from the PCB overlay diagrams, and we found that it's easiest to fit the links before adding any components.

Although we have used multi-way IDC plugs and sockets where it makes life easier, for individual connections and coax cables we use Veropins, or similar. (In some cases only the coax core gets a Veropin, and the braid is soldered directly to the earth plane.) Again, it's worth fitting the pins before you start adding other components.

And with all of that in mind, you are ready to start on the first board.

The Control Board

The control board described here is heavily based on the earlier QRP2001 controller, but with a few relatively minor hardware updates and some significant software changes. The same PIC microcontroller has been retained - the PIC16F877A. This is a 40-pin device which gives us just enough I/O lines to connect everything directly to the PIC rather than needing external latches, thus minimizing parts count.

An RS232 serial interface allows the QRP2004 to be controlled from a PC. The control protocol is very rudimentary at present, but will be expanded in future if there is any interest in that particular feature. In fact the PIC, a MAX232 for the serial interface, and a handful of support components are all that there is on the control board. A rotary encoder, keypad and LCD display module need to be connected to provide the 'user interface': more on this later.

The schematic (Fig. 3) shows just how simple this circuit board is – it's mostly connectors and decoupling capacitors!

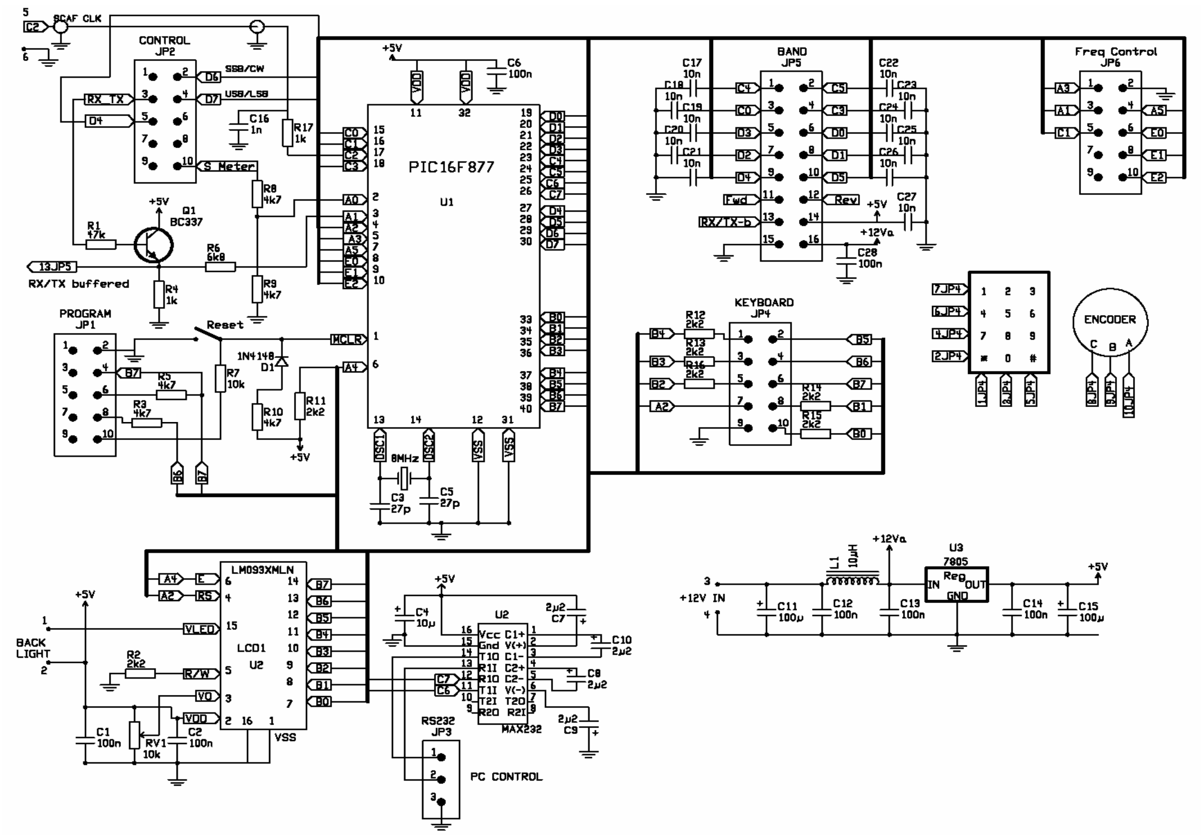

Fig. 3: Control Board Circuit

Figure 4 is a picture of one of our prototypes, which shows the connectors for the in-circuit programming interface, VFO control and system bus on the left (along with the clock crystal), the PIC slightly more to the right, and the remaining connectors and support components further right again. The 5v regulator can be seen bottom right, and this benefits from a small heatsink if you use an LCD display with a backlight.

On that subject, a backlit LCD is much easier and nicer to read under most light conditions, but a typical backlight of this type draws 100 - 200mA. For the best of both worlds, fit an LCD display with a backlight, but route the backlight supply via a miniature toggle switch on the front panel so that you can switch it off when power consumption is an important issue. Alternatively insert a resistor with a value between 10 to 47 Ohm in series the LCD backlight supply lead. This will reduce current consumption albeit with a corresponding reduction in intensity of the backlight.

The MAX232 for the serial port is a surface mount device, and (in keeping with the earlier construction notes) is underneath the board.

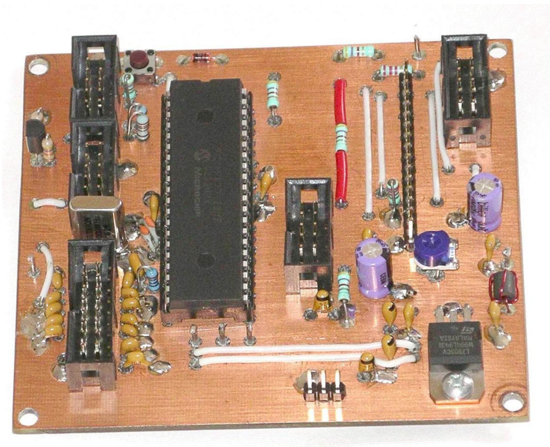

Fig. 4: Control board

We use ribbon cable for quick and easy distribution of control signals between the various boards of the QRP2004. For example, the control board is connected to the low pass and band pass filter boards by the so-called 'system bus'. This carries band selection signals, forward and reflected power indications for the ALC, and low-current power supply lines all on a 16-way ribbon cable. We used rather nice (but expensive) keyed connectors on the prototypes so that we couldn't plug things in the wrong way around. However, you *can* use just twin-row 0.1" pin strip and matching IDC connectors if you are careful to insert the plugs the right way up.

The LCD display also connects to the control board through another length of 16-way ribbon cable. However, in this case we haven't used IDC connectors, just a single-row 0.1" pin-strip on the control board which matches the corresponding pin-outs on most suitable alphanumeric LCD panels. We use a 16 character by 2 row display on the QRP2004 – for example the Epson LM032. You'll need to fan out the 0.05" pitch ribbon cable at the ends to match the pitch of the pin-strips.

The full list of external connections to the control board is:

- JP1 allows the PIC to be re-programmed without having to remove it from the control board.
- JP2 is rather under-utilized at the moment, but carries a selection of miscellaneous control lines to the audio and mixer boards.
- JP3 is the RS232 serial port which can be used to control the QRP2004 from a PC.
- JP4 connects both the keypad and the rotary encoder to the control board. Note that this connection is a single 10-way ribbon cable at the control board end, but at the other end it needs to be split into separate 7-way and 3-way strips (for connection to the keypad and encoder, respectively)
- JP5 is the 'system bus' connector, mentioned previously.
- JP6 controls the VFO, and also three of the control lines to the mixer board.

Copyright © 2004 - 2005 by QRP2004 Design Team

Apart from these connectors there is the 16-way ribbon cable to the LCD display (mentioned earlier), a 12V power connection, and a screened output (thin coax) which carries a clock signal to the SCAF on the audio board.

Since the control board circuit is so simple, we haven't provided a detailed description. However, operation of the RX/TX switching logic is worth a closer look since it is a little unusual. The PIC does not control RX/TX mode selection: there is discrete logic on the audio board which performs this function. The RX/TX line from the audio board comes into the control board where it is buffered by Q1 before being distributed to other boards through the system bus. Of course the PIC does monitor the RX/TX line so that it ‘knows’ what state the radio is in.

A few PIC pins have to perform two functions. Despite having about 30 I/O pins to play with, we still ran short. (This proves the old adage that the requirement grows to fill the capacity...and then a bit more.) In some cases lines act as outputs to two devices: this is easily managed by ensuring that only one of those devices is enabled at a time. A2 is an example of this, since it controls one of the LCD signals and one of the keypad output lines, but at any given instant only one of these is enabled. Other lines act as both input and output: B2 is both an output to the LCD and an input from the keypad. By connecting the keypad line via a resistor, the PIC can override the logic level from the keypad and hence still control the LCD correctly. However, we configure the B2 pin as a high impedance input (tri-state) when polling for keypad input, so that the keypad can now dictate the signal level.

The rotary encoder which we use for the main tuning control is a cheap Bourns mechanical encoder which has 24 positions per revolution, with a click-stop at each position. However, we want a lot more than 24 tuning steps per revolution, so we take advantage of all 4 signal transitions which the output lines go through on each 'click'. This allows us to achieve 96 steps per revolution from a cheap encoder. The catch is that you have to 'de-click' the encoder to get rid of the detents.

The specific encoder which we used is the Bourns ECWJ-B24-AC0024, and you ‘de-click’ it as follows:

- With a sharp hobby knife cut the black plastic cups at the back of the rotary encoder. (**Warning: Mind your fingers!**)
- Carefully prise open the assembly taking care not to lose the small nylon washer at the end of the shaft.
- Remove the sliding contact fingers and note how the fingers are bent with respect to the main body of the metal washer.
- Using two small pliers bend the contact fingers of the metal washer to exactly the opposite position.
- Reassemble but with the metal ring reversed so the click indent finger points towards the shaft making sure that there is sufficient contact pressure.
- Use instant glue sparingly to glue the 4 black plastic locating pins of the front assembly to the rear part of the encoder housing.

The firmware for the PIC microcontroller is available on our web site (Ref. 11) as a .HEX file and, depending on the level of interest, we may also make pre-programmed PIC processors available. However, for those who wish to amend the firmware, or are just interested, we are very happy to provide source code. The firmware can be built using tools which are freely available on the Microchip web site, and can be programmed into the PIC16F877A without even unplugging it from the control board. Again, if enough people are interested we will provide instructions on how to do this.

If you do make any changes to the source code, we would be very keen to hear about it. You will probably also want to perform a reasonably thorough test of your amended firmware. With that in mind, we have produced a test plan which we use to regression-test minor changes to the firmware, and you can download it from our web site.

Control Board Construction and Testing

Construction of the board is very straight-forward. We strongly recommend that you use a socket for the PIC processor, but all other components (apart from the display and controls) are mounted directly on to the PCB. The component list, PCB layout, and component overlay diagram are available on our web site.

Refer to the General Construction Notes above, take your time and enjoy it! Don't fit the PIC in its socket yet, or connect the LCD display panel.

Once everything has been fitted and soldered to the board, carry out the customary careful inspection for solder bridges, dry joints, etc. Then apply 12V from a current-limited power supply, check that the on-board voltage regulator provides 5V on its output, and then remove power again.

Assuming that you have a pre-programmed PIC, you are ready to start more serious testing. If you are planning to program your own PIC then we recommend that you look at WinPic by Wolf, DL4YHF (Ref. 10), unless you have already have PIC programming tools. WinPic is superb free PIC programming software for PCs running Microsoft Windows™. The WinPic help file includes details of a number of different PIC programming hardware options, and we used a circuit very similar to the design billed simply as "A programmer for FLASH- and EPROM-based PICs".

Insert the programmed PIC into its socket and connect the LCD display to the 16-pin strip on the control board. Make sure that you connect pin 1 of the display to pin 1 of the strip (the end nearer to the 5V regulator).

Once again, apply 12V from a current-limited power supply. How much current your board requires is dictated by the backlight of your LCD display (if you have a backlight). Everything else takes only a few milliamps, but as noted earlier the backlight might take 100 – 200mA.

If all is well you should see the QRP2004 display appear on the LCD, with the top row showing 7.0MHz and the bottom row displaying current operating mode and settings. Before you can actually read the display you may need to adjust the LCD contrast control. This is the only adjustment provided on the control board, in the form of preset potentiometer V1.

The next step is to connect the keypad and rotary encoder via 10-way ribbon cable to connector JP4. Note that pins 1 to 7 of JP4 go to the keypad, whilst pins 8 to 10 go to the encoder. We have noticed that the pinouts of 4 x 3 keypads from different suppliers are ordered differently – they all have 7 contacts corresponding to the 3 columns and 4 rows, but there is no standardisation (that we are aware of) on how they are ordered on the back of the keypad. Check your particular keypad with an Ohm meter and compare against the control board circuitry. For example when key "5" is pressed, 6JP4 is connected to 3JP4 and the software reads this accordingly. Getting these connections right may involve some trial-and-error!

Once the keypad and encoder are connected, you can 'drive' the front panel controls just as if there is a radio attached. Have a good play with these controls so that you are confident that the control board is working before moving on to the next module. The keypad buttons have the following functions:

- [1] Increase audio bandwidth (8 is max)
- [2] Increase tuning step size (4 is max)
- [3] Move to next memory (no effect until memories have been programmed)
- [4] Decrease audio bandwidth (1 is min)
- [5] Decrease tuning step size (1 is min)
- [6] Move to previous memory (no effect until memories have been programmed)
- [7] Toggle LSB/USB/CW mode
- [8] Store memory (set memory <n> to current freq.)
- [9] Recall memory (set frequency from memory <n>)
- [#] No effect on its own (used as 'accept' key)
- [0] Start direct frequency entry (# or * to finish)
- [*] Invoke special function (# to exit):
    - [*][0] Set S-meter zero level
    - [*][1] RIT control
    - [*][2] Calibrate DDS reference frequency

The Chassis

At this point in the proceedings it's useful to assemble the chassis so that you have somewhere to mount each board as you build it. The chassis is constructed entirely from blank double-sided PCB material, and consists of a large horizontal rectangle, surrounded by four sides, roughly along these lines of Fig. 5.

Copyright © 2004 - 2005 by QRP2004 Design Team

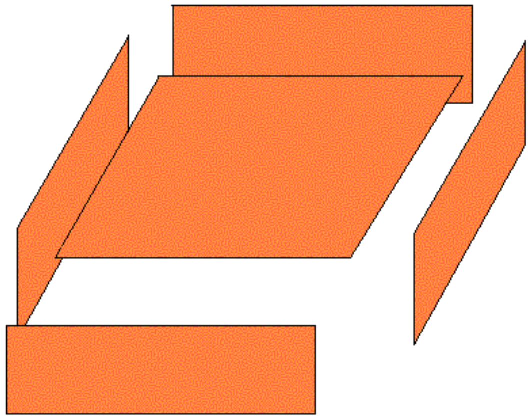

Fig. 5: Chassis Construction

The left side, right side, and rear panel are all the same height, and the horizontal plate is attached half way up each panel so that there is room both above and below to mount the circuit boards. The front section is about a centimetre shorter than the others, and is mounted so that the lower edge still aligns with the others: this leaves a gap at the top, making it easier to run the cables from the front panel controls back to the control board.

The image above implies that the front and back panel are the same length as the corresponding edge of the horizontal section. You could indeed build it like this, but we suggest that you make the front and back panel 3 – 4 cm longer than needed. This leaves 1.5 – 2 cm of overlap on each corner of the chassis which can be useful for attaching an outer cover to protect your pride-and-joy when it's all finished.

The plan view (Fig. 6) shows dimensions for the chassis which are close to the <u>minimum</u> that will accommodate the circuit boards (but allowing for some overlap on the front and back sections as noted above). The minimum height of the back and sides is 7cm (to accommodate the LPF and BPF boards), with the front panel 6cm high. In practice we recommend that you make your chassis a bit bigger in all directions so there is more room for manoeuvre:

Copyright © 2004 - 2005 by QRP2004 Design Team

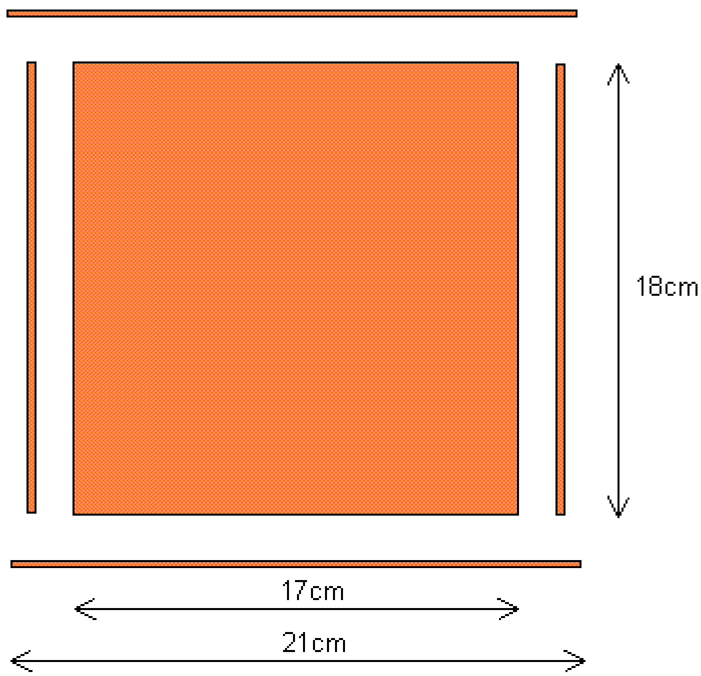

Fig. 6: Chassis Dimensions

One of the nice things about this chassis is that assembly couldn't be much easier! When you are happy that all of the panels are the right size, with right-angles in all the correct places, you just solder it together. Initially just 'tack' the sections together with spot joints. Check that the chassis stands correctly on a flat surface, and that all edges are tightly butted together. When all is well, run a continuous bead of solder along all internal angles, still maintaining the base on a flat surface.

The end result is a chassis which is surprisingly rigid and with excellent properties as an earth-plane. If you can't find a single piece of PCB material which is big enough to form the horizontal chassis member, then it's quite acceptable to use two pieces butted side-by-side. Solder several wire links between the pieces on both sides to ensure a good electrical connection. Fig. 7 shows one of our units which was built in this way.

Copyright © 2004 - 2005 by QRP2004 Design Team

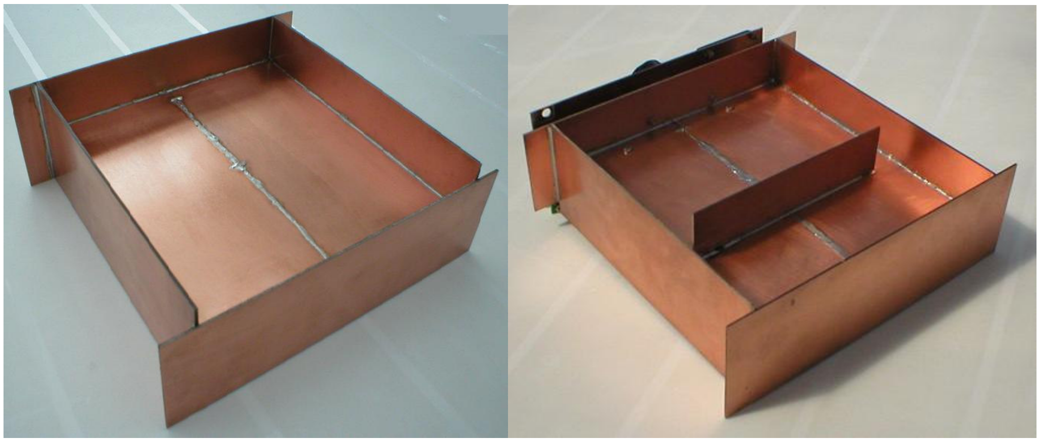

Fig. 7: Chassis Top View *Chassis bottom view (with front panel)*

The bottom view of the chassis was photographed with an early attempt at a front panel attached: please ignore this for now. The other thing which is apparent from this view is the screening section in the bottom 'cavity' of the chassis, which is intended to reduce interference between the audio and mixer boards. It may not be strictly necessary, but it's trivial to add and it's better to play it safe. The screen is just another piece of double-sided PCB material: it is the same height as the lower cavity, but about 2cm narrower so that there is a gap at each end for interconnections to pass through.

The screen is simply soldered (on both sides) to the horizontal chassis plate. Before soldering make sure that the screen is positioned so there is enough room in front of it for your audio board (which is 10cm deep), and enough room behind for the mixer board (6.5cm deep).

If you are concerned about the chassis surface tarnishing with age, then give it a couple of coats of clear varnish or lacquer. Graphite spray might also be a good alternative, although we haven't tried this. We doubt this has any effect on electrical properties, but it keeps it looking pretty. However, if you do coat the chassis with anything non-conducting <u>make sure</u> that you thoroughly clear away the coating around the PCB mounting points, so that you still get a really good electrical connection to ground.

The display and controls are not mounted directly on the chassis – they mount on a separate front panel, which then attaches to the chassis. The front panel can also be made from PCB material, or it could be aluminium … or whatever takes your fancy. Layout of controls is also a very personal thing, so we aren't going to recommend any particular arrangement. However, if you run short of inspiration, G8BTR's front panel (Fig. 8) should help.

Fig. 8: Example front panel

The main thing to consider is that you need room for the LCD display module, a tuning knob, two smaller rotary controls (volume and RF drive level), 12-button keypad, and microphone socket. You may also want some combination of headphone socket, loudspeaker socket, and/or morse key socket, although these could be at the back of the rig if you prefer.

Once you have cut your front panel to size, mount the LCD panel, keypad and rotary encoder on it. Then attach it to the front face of your chassis: again the exact method will vary with layout, but you will need to use some form of standoff pillars to leave room to accommodate the controls and LCD module. Fig. 9 shows our first prototype with a temporary front panel made from PCB material, prior to mounting the LCD module and rotary encoder.

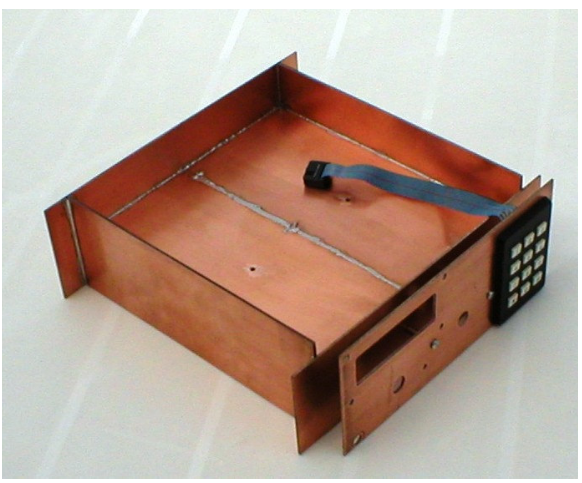

Fig. 9: Chassis with front panel attached

When your chassis is finished, and the front panel is fastened in place, mount the control board at the front of the upper section. We used M3 bolts and short (5 – 10mm) metal standoff pillars to ensure that the ground plane of the circuit board is well bonded to the earth provided by the chassis (remember to remove any coating around the bolt holes first). So now your radio should look a bit like Fig. 10.

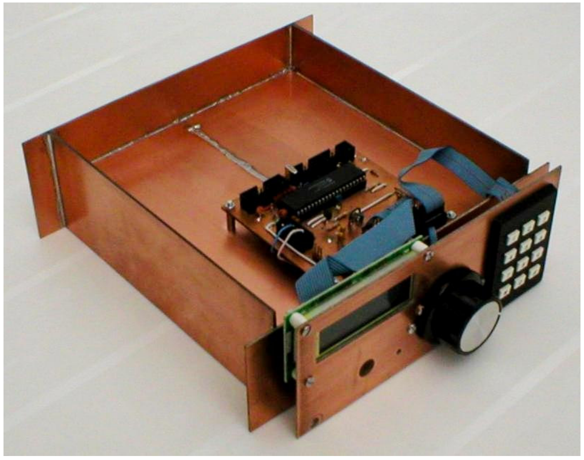

Fig. 10: Chassis with Front Panel and Control Board fitted

Reconnect the LCD panel, keypad and rotary encoder to the control board, apply power, and check that the controls all still work. If all is well, then it's time to build the VFO.

The VFO & PLL

Our VFO is heavily based on a highly-regarded design by Klaas Spaargaren PAOKSB (Ref. 5). Klaas' design covers a complete octave with a single VCO: it works brilliantly, but we found that the phase noise is rather higher at the frequency extremes (especially at the lower end) than in the middle of the range. To further improve Klaas' excellent design we use two separate VCOs, each covering half an octave.

Our mixer needs a VFO input at *twice* the frequency of interest, so to operate up to the 10m band we need a VFO that can work up to about 60MHz. In fact our main pair of VCOs cover the frequency range from 32MHz to 64MHz, corresponding to an operational coverage from 16MHz to 32MHz. How we cover lower frequencies is revealed later in the description of the mixer board.

VCO3 is optional, and covers a range from 100 to 108 MHz for those who want to experiment with coverage for the 50MHz band or run between 112 and 121.8MHz to provide full coverage for the European 80m, 40m, 20m and 10m band and partial coverage (1.8 – 1.9 MHz) of the 160m band with even higher performance.

Regardless of which VCO is active, its output signal is buffered by Q9 to Q13, divided by U7a and U3, and then routed to the PLL chip U5. The reference signal for the PLL is generated by U8, the AD9835 DDS chip. The DDS output is frequency-divided by a factor of 10 by U4 before going to the PLL, to reduce phase noise.

The frequency and phase of the VCO input and the DDS reference frequency are compared by the PLL to generate a DC-control signal which is buffered and filtered through U6 and applied to the varicap diodes of the active VCO selected. Selection of the correct VCO, and setting the correct DDS reference frequency, are both handled automatically by the microcontroller board.

The loop filter formed by U6 is taken directly from PA0KSB's VFO design. The CA3140 is noisier and has less output voltage swing than many more recent op-amps, although the effect of output noise is alleviated to some degree by the filtering from R51/C62, followed by R2/C4/R1/C1. Improved output voltage swing should make VCO alignment less critical, although we didn't find this too much of a problem with our prototypes. We plan to investigate the benefits of using a better amplifier when time permits, and if this proves beneficial we'll post an update on our web site. Meanwhile, you might like to fit U6 in a socket to make substitution of another part straight-forward.

Fig 11 is a block diagram of the VFO, and it's worth re-reading the description above with the block diagram in front of you. Fig 12 shows the corresponding circuit diagram.

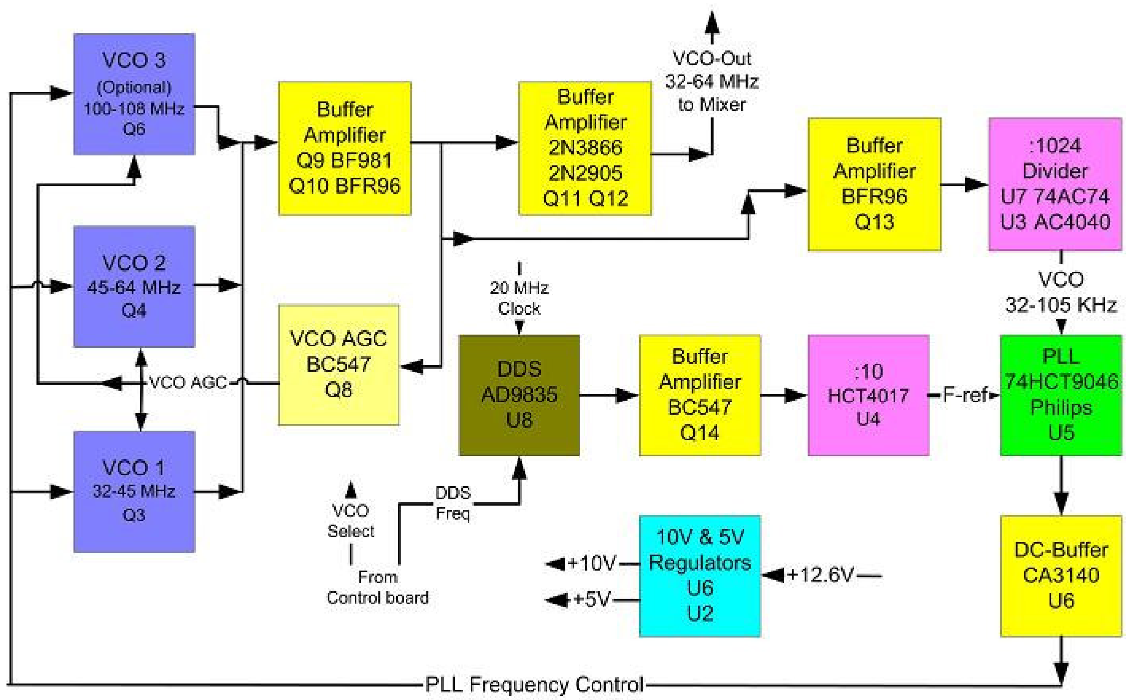

Fig. 11: VCO & PLL Block diagram

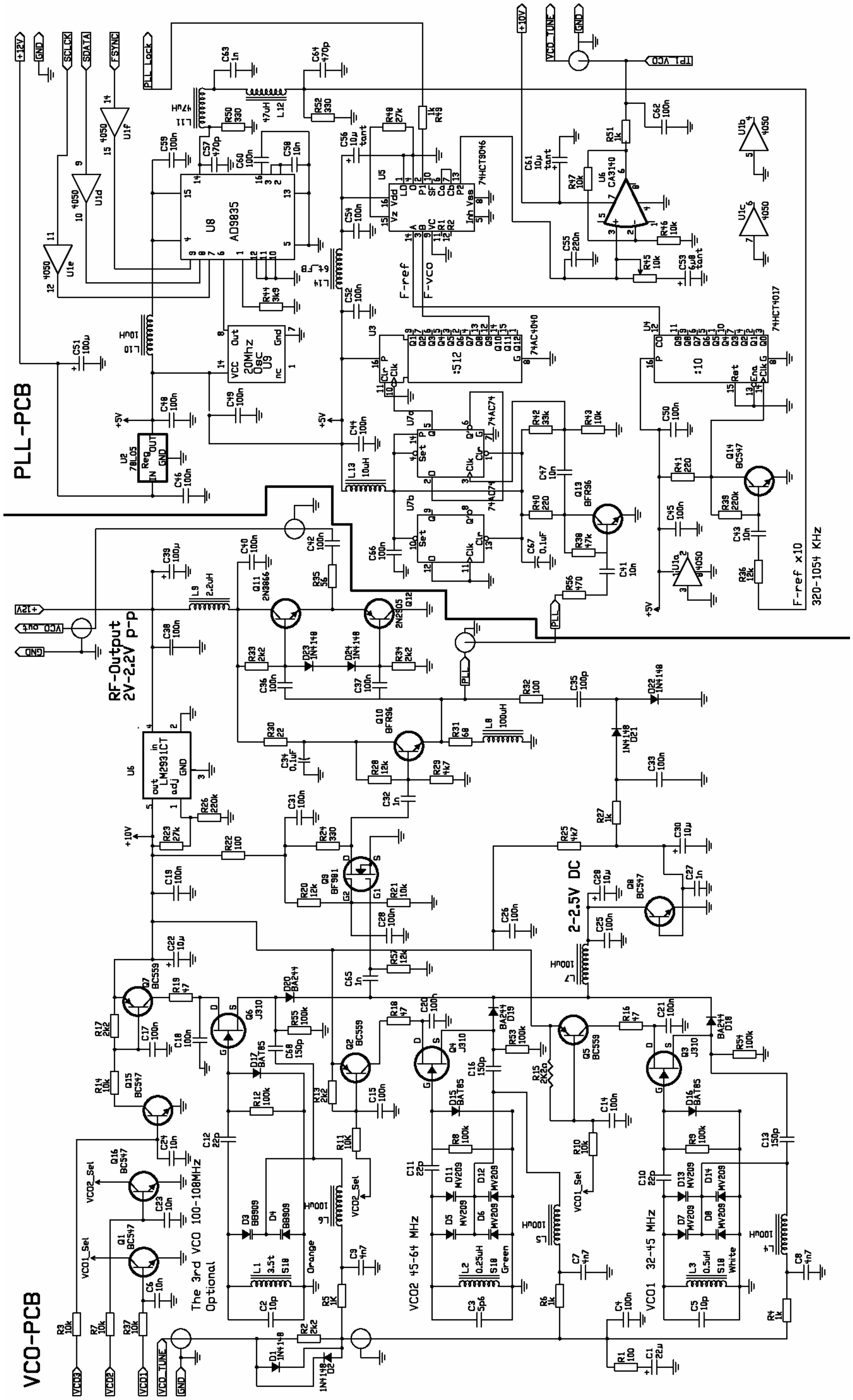

Fig. 12: VFO & PLL circuit diagram

VFO Construction and Testing

Before starting work on the PLL board, drill 4 – 5mm holes where the 'bodies' of Q10 and Q13 (BFR96) will be located. This allows the legs of these devices to lie flat against the tracks of the PCB, where they will be soldered. Take care when mounting U8, the AD9835 DDS chip as the pin spacing is only 0.65mm! Before soldering this IC clean the PCB pads first and wet with a flux pen. When soldering use the smallest soldering tip at your disposal, and we recommend using 28 swg (0.4mm) 60/40 solder. Use solder wick to remove any solder bridges between the pads. U1 and U7 are also SMD chips, however these have 1.25 mm spacing and are therefore much easier to solder on the PCB.

As noted earlier, we suggest that you fit a socket for U6, the CA3140 amplifier. You'll see from Fig. 14 that we didn't spot this in time for our own prototypes!

The VCO coils (L1, L2 and L3) are all Toko S18 types. The S18 series is colour-coded to indicate the number of turns: L1 is an S18 orange, L2 is green, and L3 is white. Although Toko have stopped making S18 coils now, they still seem to be quite easy to obtain.

We recommend fitting the optional screening cans for all three of these coils. Before soldering the screening cans for the VCO coils to the top foil, screw out the cores with a plastic trimming tool so they extend approx 6mm above the top of the coil. This will enable you to fit the Screening coils over the coils in the correct position, allowing for a different or even a second core to be screwed into the coil during alignment.

When fully built, the VCO and PLL boards should look like Fig. 13 and 14.

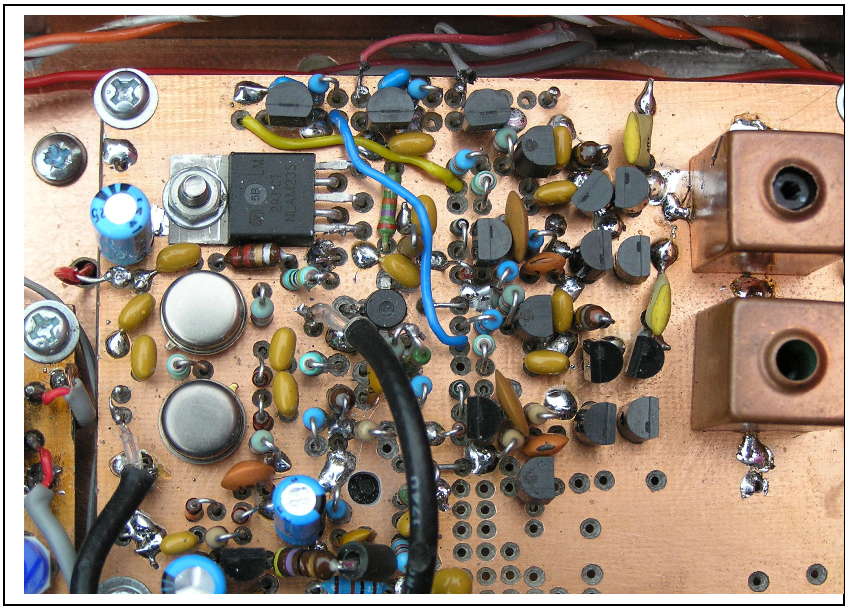

Fig. 13: VCO board – note VCO3 has not been fitted

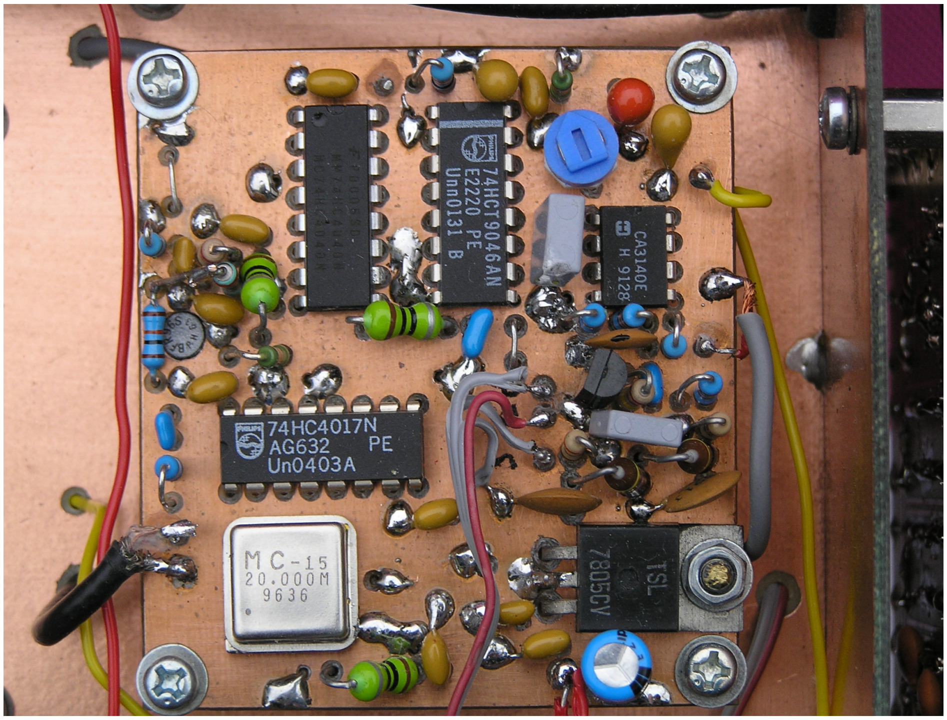

Fig. 14: PLL board: SMD ICs are fitted on the trackside of PCB

Alignment:
Connect the VCO and PLL boards temporarily to a current-limited 12V supply, and check that current consumption is <200mA. If all is well, then temporarily connect the DDS control lines (SCLCK, SDATA, and FSYNC) to the control board (JP6 pins 1, 3, and 4 respectively).

You also need to connect the VCO select lines, VCO1, VCO2 and VCO3 to the control board. We have not provided an explicit connector for this on the control board yet (the VCO select lines are a recent additional to the design) but we are planning to add this to the next revision of control board PCB. In the meantime, we suggest that you wire VCO1, 2 and 3 through to the back of the band select connector (JP5) of the control board. VCO1 connects to pin 7 of JP5, VCO2 connects to pin 5 of JP5, and VCO3 connects to pin 10 of JP5.

Do not forget the +10V connection from the VCO board to the PLL board. Use miniature RG174 coax cable to connect from VCO_out to the LOx2 connector on the mixer board. Finally a screened audio cable connects VCO-tune on the PLL board to the VCO board.

You are now ready to start alignment as follows:

1. On the PLL board adjust preset R45 near U6 until the PLL is stable. This is easily checked when the receiver is built as any AF oscillation within the PLL can be heard on the headphones or loudspeaker.
2. Connect a voltmeter, set to 12V DC range, to the VCO-tune line.

Copyright © 2004 - 2005 by QRP2004 Design Team

3. Set the receive frequency to 3995kHz and check that VCO2 line is now enabled. Using a TOKO plastic trimming tool adjust the core of VCO2 coil L2 until the DC voltage on the VCO-tune line is 7.9V.
4. Set the receive frequency to 2830kHz and check that the VCO-tune line is at 1V DC or higher.
5. Set the receive frequency to 2810kHz and check whether VCO1 is now enabled. Adjust VCO1 coil L3 until the DC voltage on the VCO-tune line is 7.9V. If this voltage cannot be reached then the inductance of L3 needs to be increased. This is easily achieved by setting the original core until it is flush with the bottom of the coil and fitting a second spare S18 core in L3.
6. Set the receive frequency to 2010kHz and check that the VCO-tune line is >1V DC.

Trouble-shooting:
`

If you cannot get the VCO and PLL to lock then check the following:

- Reset the microcontroller by pressing the "reset" pushbutton on the control board.
- Set the receive frequency via the keypad to 3500kHz
- Connect an oscilloscope to the output of the DDS Low Pass Filter (junction of R52, C64 and L12). You should measure a sinewave of at least 0.4V peak to peak. Connect a frequency counter at this point: it should indicate a frequency of 546.87kHz.
- Next connect the scope to Pin 14 of U5 (74HCT9046) PLL chip. This is the reference input where you should see a 5V squarewave, and measure a frequency of 54.68kHz
- Repeat the measurements on the VCO output to the PLL board. RF voltage should be approximately 2V peak to peak sinewave with a frequency of 56,000kHz.. If this signal is present, check whether U5 pin 3 shows a 5V squarewave with a frequency of 54.68kHz.

The Mixer

Strictly speaking, this should probably be referred to as the 'product detector' since we are converting directly from RF to audio/baseband frequencies. However we've adopted the rather sloppy habit of using these two terms interchangeably in the QRP2004.
We first encountered this particular mixer/product detector through the work of Dan Tayloe. We have since seen it referred to as a QSD (Quadrature Sampling Detector), and there has been some discussion in amateur radio circles recently over the 'real' origins of this circuit topology. However, in our minds it continues to be the Tayloe Product Detector.

In the QRP2004 we use a quasi-balanced form of mixer where the two sections of a QS3153 dual switch are both driven 'in opposition' to each other. This helps to cancel out most of the digital switching artefacts. In practice we found that this makes little or no difference when receiving (i.e. down-converting) but is crucial when transmitting (up-converting). This variant of QSD was brought to out attention by Gerald Youngblood in his work on the ground-breaking SDR-1000 transceiver (Ref. 6).

We use two of these circuits on our mixer board, one for receive and one for transmit. At any given instant only one circuit is enabled, and the other is in a high-impedance state.

The strong signal performance, great selectivity and modest drive requirements of these circuits are major contributors to the performance of the QRP2004.

The mixer board schematic is shown in Fig. 15.

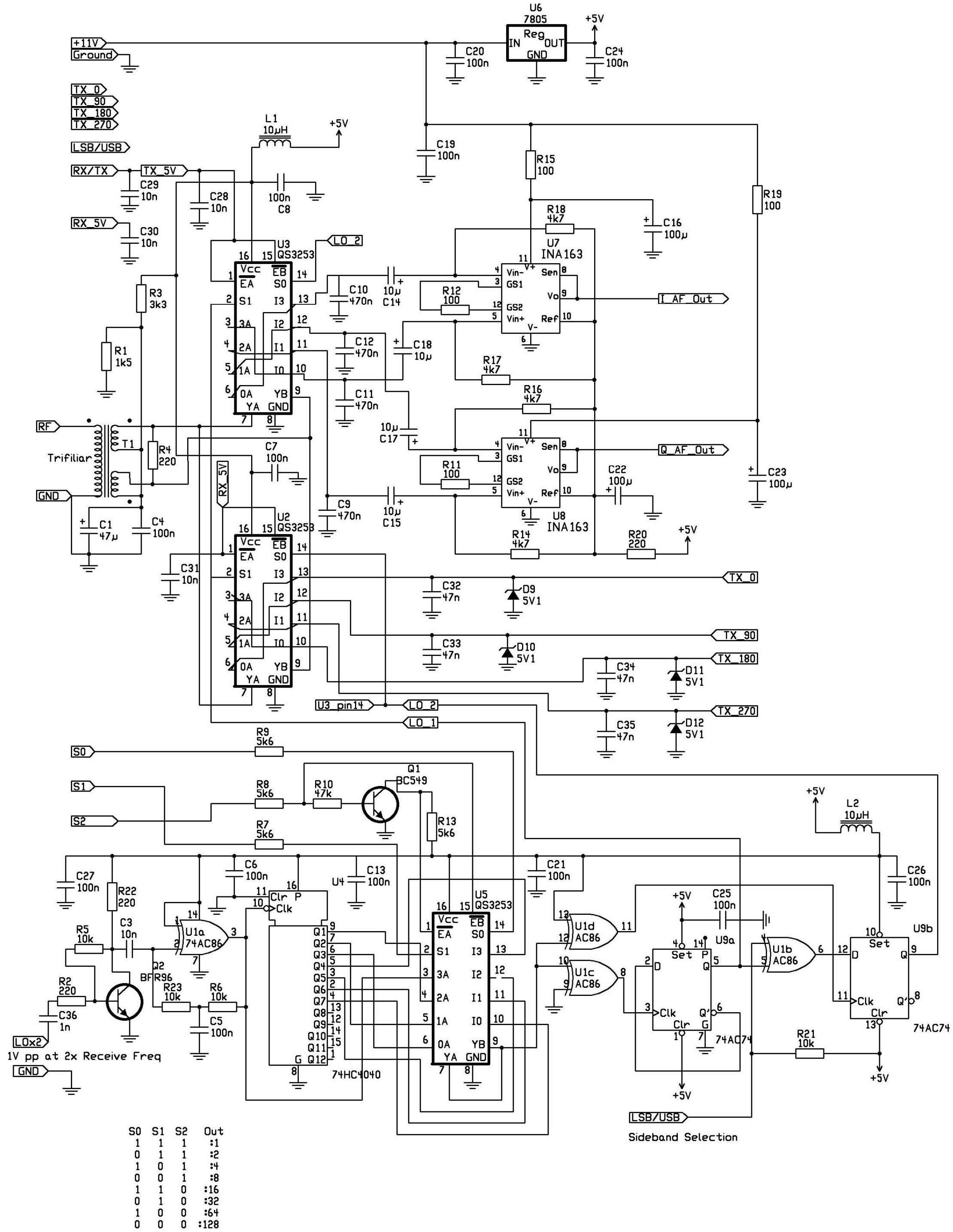

Fig. 15: Circuit of Mixer Board

The RX and TX mixers are shown top-right of the schematic. The RX mixer is followed by two INA163 low-noise instrumentation amplifiers, and their outputs provide the 'I' and 'Q' quadrature audio signals to the AF board whilst receiving.

Bottom right is the drive circuitry which converts our VFO signal (at twice the frequency we want to receive or transmit) into two squarewave signals in quadrature at the required signal frequency.

Bottom left is U4 a 74AC4040 divider, which divides down the raw VFO signal across eight octaves. U5 selects the required output, under microprocessor control through control lines S0, S1 and S2 and routes the VFO signal to U1c and U1d. Since the VFO can tune across one whole octave, and the mixer can subdivide down to lower octaves, we can achieve complete general coverage across an eight-octave range of the HF and LF spectrum. Generation of quadrature LO signals is performed by U9a and U9b which produces two squarewaves, LO_1 and LO_2 to drive the Tayloe mixer. Sideband selection is done through U1b, which inverts the signal from U9a pin 5 under microprocessor control, resulting in LO_1 lagging instead of leading LO_2 by 90 degrees.

The divider was originally located on the VFO board (where it logically belongs). However, we found that the digitals signals around this part of the circuit are prone to noise pickup, which appeared as phase-noise at the mixer. Relocating the divider to the mixer board helps to reduce this effect.

Mixer Construction and Testing

Before starting work on this board, drill a 4 – 5mm hole to locate the 'body' of Q2 (BFR96), in a similar manner to the PLL board. In all other respects, construction follows similar lines to all of the other boards. Fit the earth links and other wire links first, then all of the leaded components (but not the SMD parts yet).

It is important that the capacitors to ground on the 'receive' mixer (C9 – C12) are of high quality. They are effectively integrating wide-band RF from the switches, such that signals either side of the switching frequency integrate to zero over time, whilst the narrow band of signals we want integrate to non-zero values varying at audio frequency.

Of course, in most cases we are trying to pull very low levels of wanted signal from high levels of wide-band 'background noise'. To achieve this difficult trick we need capacitors with low equivalent series resistance (ESR) and low inductance. Polypropylene dielectric would probably be ideal, but would be quite large for the necessary values. In practice we found that good quality polyester capacitors work pretty well: polycarbonate may also be suitable be we haven't tried that.

RF chokes L1 and L2 each consist of 6 turns of thin enamelled wire (approx. 30 SWG) on an FX1115 (or similar) ferrite bead. Transformer T1 has a single trifilar winding of 12 turns

Copyright © 2004 - 2005 by QRP2004 Design Team

(also 30 SWG enamelled wire) on an FT37-61 toroid core. Be careful to observe the correct phase of each conductor in the trifilar winding, as shown in the schematic, when wiring up T1.

When all the leaded components have been installed, solder in the INA163 amplifiers on the track-side of the board. Apply power to the 12V input from a current-limited power supply, and check that 5V is provided by the on-board regulator. The output voltage from both INA163s (i.e. at the I and Q output connectors of the board) should also be very close to +5V. Q2 collector voltage should be between 2 and 3 volts, and total current consumption for the board should be about 30mA at this stage.

Remove power and, assuming that all is well, fit the remaining SMD ICs. You can substitute FST3253 switches for the QS3253 devices shown on the schematic if these are easier to obtain. Mount the mixer board at the back of the lower chassis area with M3 bolts and short (5 – 10mm) metal standoff pillars, so that your rig looks like Fig. 16 when it is upside-down.

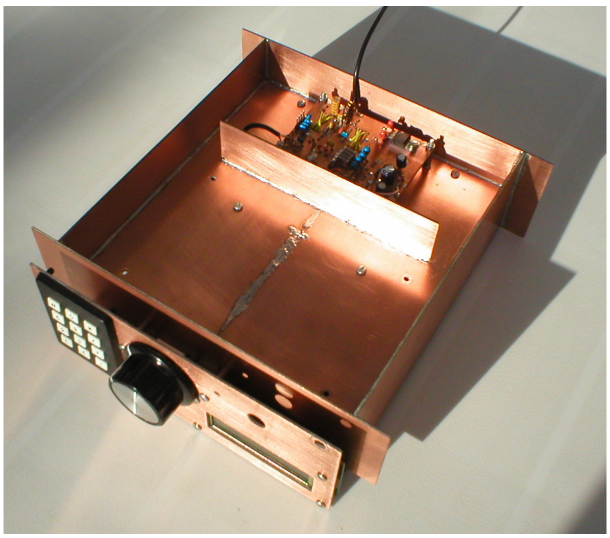

Fig. 16: Mixer Board mounted in chassis

To test the mixer, you need to connect the output from the VFO to the 'LO x 2' input of the mixer via thin 50Ω coax. Drill a small (approx. 3mm) hole through the main chassis plate to get the coax from the upper 'cavity' to the lower area where the mixer is mounted. Temporarily hook up a convenient antenna connector to the 'RF' input of the mixer, again with thin coax.

You will also need to make some connections to the control board in order to complete installation of the mixer. The three octave-select signals (S0, S1 and S2) are provided by JP6 of the control board. Attach about 12" of ten-way ribbon cable to a suitable IDC connector, and plug into JP6 on the control board. However, only some of these lines go to the mixer: others go to the VFO module, so the ribbon cable needs to be split in half. Conductors 1 – 5 go to the VFO, to replace the temporary connections which were installed to test the VCO and PLL boards. Now you need to drill a hole through the horizontal chassis plate so that conductors 6 – 10 can be routed to the mixer board.

Copyright © 2004 - 2005 by QRP2004 Design Team

Once it is in place, cut the ribbon cable to length and connect conductors 6, 8 and 10 to connection points S0, S1 and S2 (respectively) of the mixer board. Conductors 7 and 9 are just trimmed off.

The USB/LSB control line also needs to be wired through from the control board. Pin 4 of JP2 on the control board should be connected to the USB/LSB connection point on the mixer board. Although JP2 on the control board is a 10-way header designed for IDC ribbon cable connectors, we just used insulated hook-up wire to connect up the particular pins that we need.

Finally, connect the 'TX_5V' point to ground and the 'RX_5V' point to the +5V rail. For the latter, you can temporarily attach a length of hook-up wire to the 5V output of the mixer board voltage regulator.

You now have the beginnings of a receiver. Apply power to the control board, VFO boards, and mixer board, and connect an antenna to the mixer board RF input. Low level demodulated audio should be available at the mixer "I_AF" and "Q_AF" outputs. To hear it you will need some form of audio amplifier – for example, a powered speaker from a PC. With this make-shift arrangement you should be able to tune around the bands and hear some of the stronger signals quite clearly.

VCO & Mixer Control Information					
Receive Frequency	**:N**	**S0**	**S1**	**S2**	**VCO 1/2 Switch-over**
125 kHz -249.99 kHz	:128	0	0	0	176.25 kHz
250 kHz - 499.99 kHz	:64	1	0	0	352.5 kHz
500 kHz - 1 MHz	:32	0	1	0	705 kHz
1 MHz - 1.9999 MHz	:16	1	1	0	1410 kHz
2 MHz - 3.9999 MHz	:8	0	0	1	2820 kHz
4 MHz - 7.9999 MHz	:4	1	0	1	5640 kHz
8 MHz - 15.9999 MHz	:2	0	1	1	11280 kHz
16 MHz - 31.9999 MHz	:1	1	1	1	22560 kHz

Table 1: Relationship between Receive Frequency and PLL and Mixer Control

The Audio Board

The audio board is the real engine-room of the QRP2004, providing all of the gain when receiving, as well as extensive signal processing during both receiving and transmitting. Not surprisingly, it is one of the more complex boards in the radio.

It should be quite possible to replace the entire audio board with an all-digital equivalent: either a PC with a soundcard or a separate DSP board mounted in the QRP2004. We are looking into this as a possible future development, but it is not the approach which we've adopted for the current project.

Fig. 17 is a block diagram of the audio board, which outlines all the functions performed by this board.

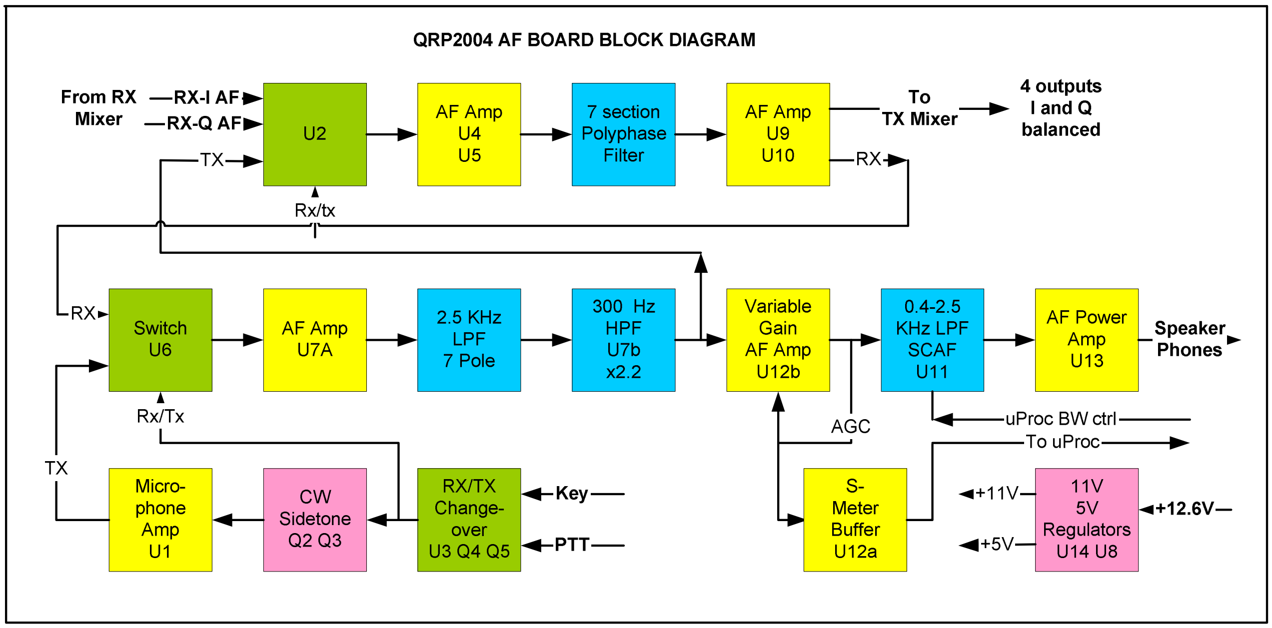

Fig. 17: AF Board – Block diagram

Common Stages: A number of stages are shared in both receive and transmit mode, with switch-over handled by U2 and U6 (coloured green). A single 7 section polyphase filter (Top row) and the 2.5kHz 7 Pole AF Filter and 300Hz High Pass Filter (U7b) are common to the RX and TX paths.

On Receive: the I and Q signals from the RX Mixer are buffered by U4 and U5, ready to drive the 0 - 180 and 90 - 270 degree inputs of the polyphase circuit. The I and Q inputs carry both the wanted and the unwanted sideband from the received signal, but their relative phase is different on each input. The polyphase network largely cancels out the unwanted sideband, leaving only the wanted signal (the unwanted signal is typically attenuated by 40 – 50dB). The gain (R37) and phase (R31) of U4 and U5 are adjustable separately to maximize sideband suppression.

The four outputs from the polyphase network are buffered by amplifiers U9 and U10. In receive mode, the 0 and 90 degrees outputs are summed by R122 and 123 (4.7kΩ) resistors to generate the AF output, which is routed via switch U6 to unity-gain buffer U7a through a passive 7 pole elliptical low pass filter courtesy of KK7B with the cut-off frequency scaled to

Copyright © 2004 - 2005 by QRP2004 Design Team

2.5 kHz. With a measured 6dB-60dB shape factor of 1.7 this filter provides a large part of the receive selectivity. U7b amplifies the output of the LPF by 2.2 to compensate for the insertion loss and also acts as a 3 pole 300Hz High Pass filter. This filter contributes significantly to the clarity of the QRP2004 as it removes 1/f noise and low frequency rumble.

With the bulk of the filtering done, the RX signal is routed to U12b, the main AF amplification and AGC stage with a control range of some 60-70dB. Presets R143 and R145 are used to set up the threshold and range of the AGC stage. U12a buffers the AGC control voltage ready to go to the Control Board, where it is used to provide an S-meter reading. U11 is an 8 pole elliptical audio filter (Switched Capacitor Audio Filter, or SCAF). The cut-off frequency of this filter is governed by its input clock frequency, which in turn is determined by the microcontroller on the main control board. By changing the clock frequency between 90kHz and 260kHz, the actual cut-off of this stage varies between 900Hz and 2.6kHz (AF cut-off frequency = input clock frequency / 100).

The AF output stage (U13) is a simple LM386 which delivers around 0.6 watts, consistent with low power consumption for portable operation. If more audio is required then this stage could be replaced by a TDA2003, which will deliver 3 watts into a 4Ω speaker.

On Transmit: U1 is the microphone AF stage, which has a preset control for adjusting the microphone gain and provision for some soft clipping. In CW mode a 700 Hz AF signal is generated by Q2 and switched through Q3 to drive to the microphone AF stage. The transmit AF signal is routed through switch U6 and filtered by the 2.5 KHz low pass and 300 Hz high pass filters. This signal is then switched by U2 which connects both I and Q inputs to U4 and U5 in parallel. The transmit AF signal is then buffered by U4 and U5 which drive the polyphase network. (See also Radcom July 2001 TT page 62 fig. 3)

The four outputs of the polyphase network correspond to the input signal, but shifted by 0, 90, 180 and 270 degrees. All four outputs are buffered by U9 and U10 and routed to the transmit mixer on the mixer board. The gain of U9a is fixed, however the gain of U9b, U10a and U10b is adjusted via presets R105, R101 and R104 for maximum sideband suppression on transmit.

Polyphase Network

The values of the polyphase network shown in Fig 19 are based on work done by JA1KO (Ref. 7). This minimizes insertion loss to less than 2dB yet uses standard component values. If desired alternative values could be used whereby all capacitors have a value of 10nF with corresponding standard resistor values ranging from 4.7K to 47K. Potential sideband suppression is identical as shown in Fig 18, however insertion loss is about 8dB higher. In practice with a 7 element polyphase network and using selected components, the limiting factor on unwanted sideband suppression is the accuracy of the quadrature LO signals rather than the performance of the polyphase network used. An excellent reference on polyphase networks is a recent article in Radcom by GJ3RAX (Ref. 7a)
Note: Make sure you feed the polyphase networks correctly, i.e. from the low -impedance stage, otherwise insertion loss will be in excess of 20dB and your receiver will be deaf.

Copyright © 2004 - 2005 by QRP2004 Design Team

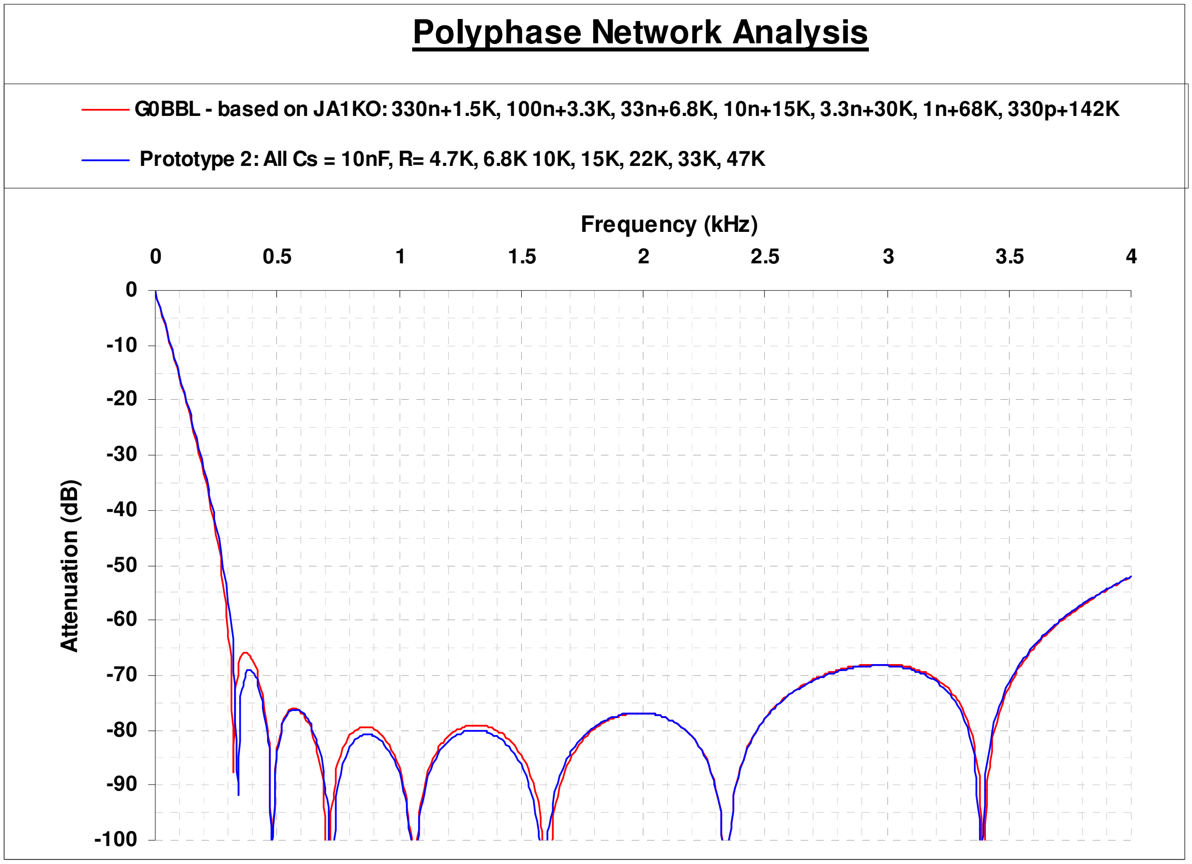

Fig. 18: Theoretical performance of 2 Polyphase networks

If you have a PC with Microsoft Excel installed, and you would like to see how the actual values of the polyphase component batches affect theoretical sideband suppression, you can try out different values with the Excel Macro (Ref. 8) which we used for our original development work.

The full schematic for the audio board is shown in Fig. 19.

Copyright © 2004 - 2005 by QRP2004 Design Team

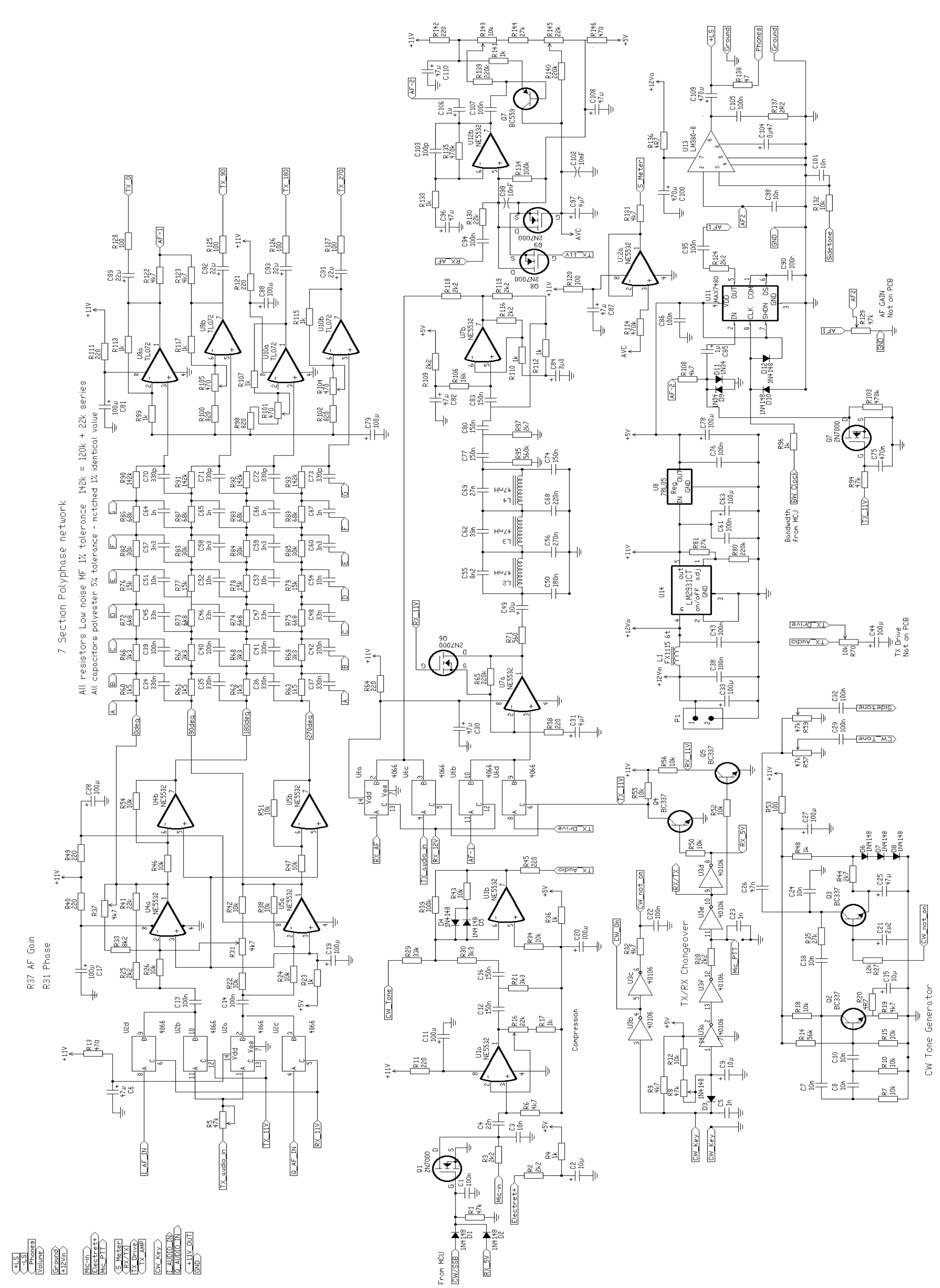

Fig. 19: QRP2004 Audio Board schematic

Copyright © 2004 - 2005 by QRP2004 Design Team

Audio Board Construction and Testing

Construction follows similar lines to all of the other boards. Fit the earth links and other wire links first, then all of the leaded components. We recommend that you fit sockets for all of the ICs (apart from voltage regulators and SMD parts). In fact there is only one SMD part on this board, U11: as usual, mounted on the track side of the board.

It is important to use matched components in the polyphase network: the absolute values of the resistors and capacitors are less important than having close-matched values within each section of the filter. We used 1% Resistors and 5% capacitors, the latter having been selected to batches of 4 components of 0.2% variation in value within the batch, using a simple capacitance meter. If you don't have access to such a meter, then at least try to use capacitors from the same manufacturing batch.

The AF board is not particular complicated to build, however as it contains more components than the other boards, the potential for errors is significant greater. Therefore we recommend you do a full check prior to installing and commissioning this board as this will pay off in terms of reduced trouble-shooting at a later stage. After mounting the audio board a number of connections are required as shown on the Interconnection Diagram in fig 20: start off with Ground and 12V Power connection and proceed with all the wiring to the mixer board. Please note that mixer board obtains the 11V supply from the regulator on the audio board, so remove the wire to +12V which you previously installed.

Next are the RX/TX, CW/SSB and S_Meter wires to the controller board. Use thin RG174 coax for connecting the SCAF clock signal from the audio board to controller board. Screened audio cables are required for all connections to various connectors (microphone, loudspeaker, headphones,) and to the two potentiometers (AF Gain and TX Drive). After a further check of the interconnecting wiring, power should be first applied from a current limiting PSU. Verify current consumption is less than 200mA before carrying out the AF board alignment procedure, which is described further down.

Copyright © 2004 - 2005 by QRP2004 Design Team

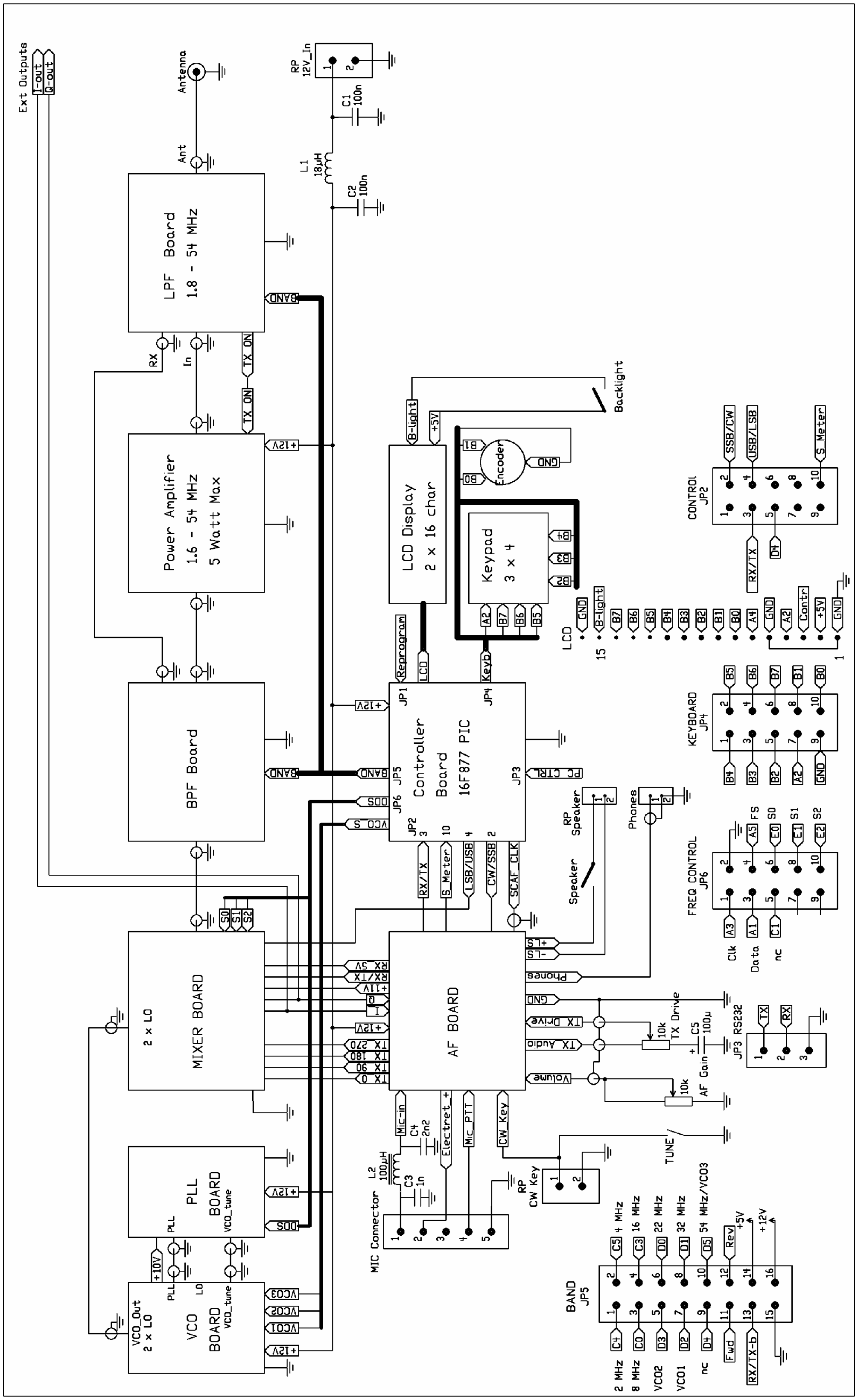

Fig. 20: QRP2004 Interconnection Diagram

Depending on the DC output voltage from your power supply, you may find that the 11V regulator, U14, gets quite hot. It's worth adding a small heatsink (for example, a simple aluminium bracket to the chassis) to keep it cool.

Band Pass Filters

The band pass filter board performs a number of very useful functions in the QRP2004. On receive it reduces the amount of wideband noise reaching the mixer, and in particular it attenuates signals from bands below the required signal. This latter point is significant with the Tayloe product detector, because it passes sub-harmonics of the required signal frequency with rather less attenuation than we would ideally like. We measured about 45 - 50dB attenuation of the first sub-harmonic on one of the prototypes, and whilst the filters only improve this by a further 5 – 10dB, it all helps.

On transmit the BPF cleans up the signal from the Tayloe product detector (which is being driven 'backwards') before it reaches the PA stage. This avoids wasting power amplifying signals which are then taken out by the final low-pass filter. More importantly, the Tayloe product detector is acting as a switch-mode mixer now, and there are some very fast edges on the output signal which might cause distortion and perhaps even instability if they were allowed to reach the PA.

There are five separate filters on the BPF board, with an optional sixth filter for the 6m band. The 160m band is actually handled by a low pass filter, rather than a band pass arrangement. The 80m band has its own band pass filter, but higher HF bands have to 'share' filters.

The correct filter for the current operating frequency is automatically selected by the control board. Filter selection and power supply lines are all provided by the 16-way System Bus ribbon cable. Indeed, the System Bus is the only connection to the BPF board apart from the RF input and output signals, which are connected via coax, of course.

The schematic is shown in Fig. 21, with the filters and filter switching at the centre of the diagram, and the PA driver and ALC circuit to the right.

Copyright © 2004 - 2005 by QRP2004 Design Team

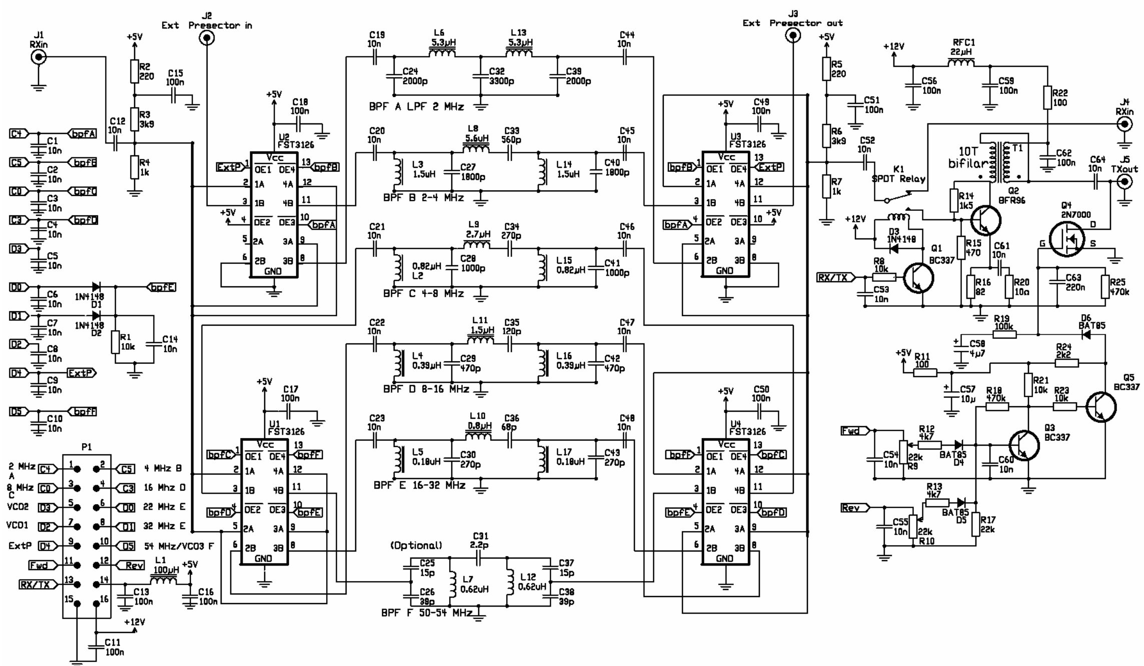

Fig. 21: BPF Circuit Diagram

The low pass filter for 160m is a simple 5 element Chebyshev design, with 50Ω input and output impedances. The component values are taken directly from the tables towards the back of the ARRL Handbook – a veritable goldmine of useful reference data and certainly much easier than calculating the values yourself!

The five bandpass filters were modelled as a 3-pole Chebychev design using an Excel spreadsheet. Nominal terminating impedance is 50Ω and component values have been rounded to nearest standard values for those who would like to use standard value axial RF chokes instead of winding T37 toroids as we did. The lower Q of these chokes have little or no impact on insertion-loss of the filters.

Since the filters only need to handle relatively small signals, we use solid-state CMOS switches to select the desired filter section. The switches we have adopted have a very low ‘on’ resistance and handle up to 3V peak-to-peak signals with very low distortion, resulting in good IMD performance.

There are really three other little circuits on the BPF board. The solitary relay, together with Q1 performs the necessary signal routing to support RX/TX switching. Q2 provides about 14dB of RF gain during transmit, to make sure that we have plenty of drive available for the PA stage. The remaining circuitry, consisting of Q3, Q4 and Q5 form the ALC circuit. Q3 starts to conduct when the forward or reflected power get above preset limits. This switches off Q5, which (in turn) starts FET Q4 conducting. Q4 shunts some of the output signal to ground, so restoring the RF output level to the desired value.

Copyright © 2004 - 2005 by QRP2004 Design Team

One final thought on the BPF board before we move on to construction details: we're eager to encourage others to experiment with the QRP2004, and improvements in the band pass filters could be particularly beneficial to overall performance. One of our prototypes was built with very narrow band-pass filters, which only covered the amateur bands. Unfortunately this restricts the rig to just six bands, and no general coverage receive option (without a major re-design).

Another alternative is to stay with the current octave filter arrangement but to use more carefully optimised filters: in particular, filters with lower in-band loss but which still maintaining fairly steep skirts. To encourage experimentation, the BPF circuit diagram shows connections for an optional external preselector, which in a future firmware revision may be selected from the keypad. The authors would be particularly interested to hear from anyone who would like to try this.

BPF Construction and Testing

As with the other PCBs for the QRP2004, this is a double-sided board with one side acting as the ground plane. Before starting work on this board, drill a 4 – 5mm hole where the 'body' of Q2 (BFR96) will be located: see the section on the mixer board for more details. As usual, fit any earth links first, and countersink other holes on the earth-plane side of the board unless that lead is intended to be connected to ground. There are some quite long links on this board to distribute the band select signals (2 links per band) out to the switch ICs: use insulated hook-up wire for these.

As always, the component list, PCB layout, and component overlay diagram are available on our web site.

Component layout on the BPF board is pretty compact, so take care over construction and build one filter at a time. Due to space constraints, use small ceramic disk capacitors for all of the filter sections. Although these are generally only available in quite broad tolerance ratings, we didn't find this to be a problem in practice. However, if you have access to a capacitance meter then it is probably worth hand selecting capacitors which are close to their nominal value.

All of the filter coils are wound on small Amidon toroids using 24 SWG enamelled wire: Table 2 shows the core type and number of turns for each coil.

Band	Coil	Type	Turns	Coil	Type	Turns
160m	L6, L13	T37-2	36	N/A		
80m	L3, L14	T37-2	19	L8	2xT37-2 stacked	26
40m	L2, L15	T37-6	16	L9	T37-2	26
30m & 20m	L4, L16	T37-6	11	L11	T37-6	22
17m – 10m	L5, L17	T37-6	7	L10	T37-6	16
6m (option)	L7, L12	T37-6	14	N/A		

Table 2: Winding details for BPF coils

In all cases the coil windings should be distributed evenly around approximately 80% of the circumference of the toroid.

T1 consists of 2 x 10 turns of 28 SWG enamelled wire wound bifilar on an FT37-43 ferrite toroid.

When you put it all together, your BPF board should look rather like Fig. 22. You'll see that we have used epoxy resin to hold the toroids in place, mainly for mechanical resilience. The switch ICs, being SMD parts, are out of sight on the other side of the board.

Fig. 22: BPF Layout

After the usual visual checks, it's time to try it out. At this point you need to make up the 'System Bus' ribbon cable. You'll need a length of 16-way ribbon cable, and three 16-way IDC sockets. The only difficulty here is working out how long the cable needs to be, and this is easier after you have mounted the BPF board into its correct place on the outside of the left-hand face of the chassis. As with the control board, mounting is accomplished with M3 bolts and metal spacers, although shorter spacers are recommended here to avoid making the radio wider than it needs to be.

Once the BFP board is in place, work out how you are going to arrange the ribbon cable to run from the BPF board, up across the top of the chassis, via the relevant 16-way connector on the control board, and then on again to the LPF board. Note that the LPF board is mounted on the opposite outside face of the chassis. Fig. 23 shows the cable which we made for one of our prototypes, whilst Fig 24 shows the complete prototype with the cable in place:

Fig. 23: 'System Bus' cable assembly

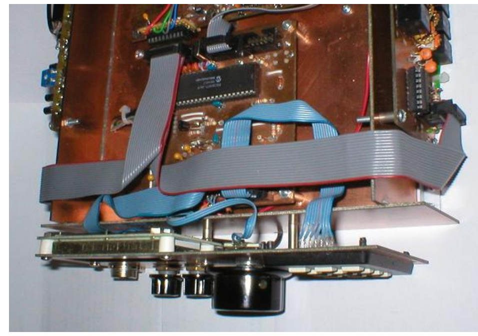

Fig. 24: 'System Bus' cable fitted

Fit the middle IDC socket in place for the control board connection. Ensure that the identifying stripe on the cable goes to pin 1 of the control board connector, and note that the cable needs to double back on itself at that connector. Leave plenty of ribbon cable either side for the BPF and LPF connections – it's better to be wasteful than have to remake the whole assembly if you find it doesn't quite reach!

Plug the cable assembly on to the control board, and dress the ribbon cable towards the BPF board. Remember that the stripe on the cable needs to end up at pin 1 of the BPF connector. When you are happy with the cable layout fit the IDC socket for the BPF board, but be careful as these connectors are very difficult to move after they have been crimped onto the cable. Don't try to fit the LPF board connector yet – leave that end of the ribbon cable un-terminated for the moment.

Double-check that pin 1 of the connector on the control board is connected to pin 1 of the connector on the LPF board. Then apply 12V to the control board – ideally from a current-limited PSU to reduce the risk of black smoke. A current limit of 300mA should be sufficient.

Assuming that all is well, you now need to confirm that the filters and associated switching are working as expected. For the BFP board the only way to really check this is to measure the filter response for each filter in turn. At this point the board should be in 'RX' mode, since there is nothing connected to the RX/TX control line yet. So connect an RF signal generator to the J1 coax connection point on the board, and monitor the RF signal level at the J4 connection point – either with an RF probe or an oscilloscope.

Set the displayed frequency to 1MHz from the keypad (press '0', '1', '0', '0', '0', '#'). Then sweep the signal generator from 1MHz up to about 3MHz, and watch for the point where the output voltage falls to half of its starting value – this is the -6dB point on the filter curve.

Copyright © 2004 - 2005 by QRP2004 Design Team

Select the next filter by setting a displayed frequency of 3MHz, and sweep the signal generator frequency either side of 3MHz looking for the upper and lower -6db points. Repeat this general sequence with displayed frequencies of 6MHz, 12MHz and 24MHz (also 52MHz if you have built the optional 6m filter). You should obtain a set of filter cut-off frequencies similar to those in Table 3, which shows the measured cut-off frequencies for one of our prototypes.

Displayed Frequency	Nominal frequency response	Lower -6dB point	Upper -6dB point
1MHz	0 – 2MHz	N/A	2.3MHz
3MHz	2 – 4MHz	1.9MHz	4.4MHz
6MHz	4 – 8MHz	3.6MHz	8.2MHz
12MHz	8 – 16MHz	7.8MHz	16.5MHz
24MHz	16 – 32MHz	14.8MHz	30.9MHz

Table 3: Cut-off frequencies for prototype BPF board

Although the exact cut-off frequencies are not critical, the aim is for each filter to cover the range shown in the table above without too much attenuation. However, priority has been given to avoiding significant attenuation in the amateur bands – we tolerate higher attenuation at the extremes of each range to ensure that we get optimum rejection of unwanted frequencies.

If you find that any of your filters show a response which is wildly different than expected, you may have a problem with the filter switching. Trace the filter selection signals through from the control board to ensure that the correct filter is being switched in.

Even when the filter switching is working perfectly, we found from building the prototypes that some of the coils may need adjustment to get the desired filter responses. It would seem that toroid cores of (apparently) the same type from different sources can vary significantly in permeability – up to 20% by our estimation. You can vary the inductance of these coils to some degree by spreading out or bunching up the windings, but occasionally you may find that you need to add or remove a winding from one or more coils to get the desired responses.

There isn't much else on the BPF board which you can conveniently test at this point. You could put the radio into transmit mode, and check that you hear the BPF RX/TX relay activate, but in general further testing is best left until later. Set both of the preset potentiometers on the BPF board (which relate to the automatic level control circuitry) to their mid-point for the moment.

Finally, connect J1 on the BPF board to the RF input/output connector of the mixer board with thin 50Ω coax. J2 and J3 are not used in this design: they are provided to support an optional external filter, but that is beyond the scope of this article. The other two RF connection points on the BPF board (J4 and J5) have to wait until the power amplifier and LPF boards are ready.

RF Power Amplifier

The little 5W (nominal) PA which we've adopted for the QRP2004 is heavily based on the excellent "QRP-PA" design from Peter Zenker, DL2FI. The original design is available as a very nice kit from Funkamateur in Germany (www.funkamateur.de), and if you have had enough of making your own PCBs by now, there's no reason why you couldn't use one of these kits for your QRP2004.

However, for the truly die-hard homebrewer we have produced our own PCB layout, and introduced a few minor circuit changes along the way. Our circuit is shown in Fig. 25.

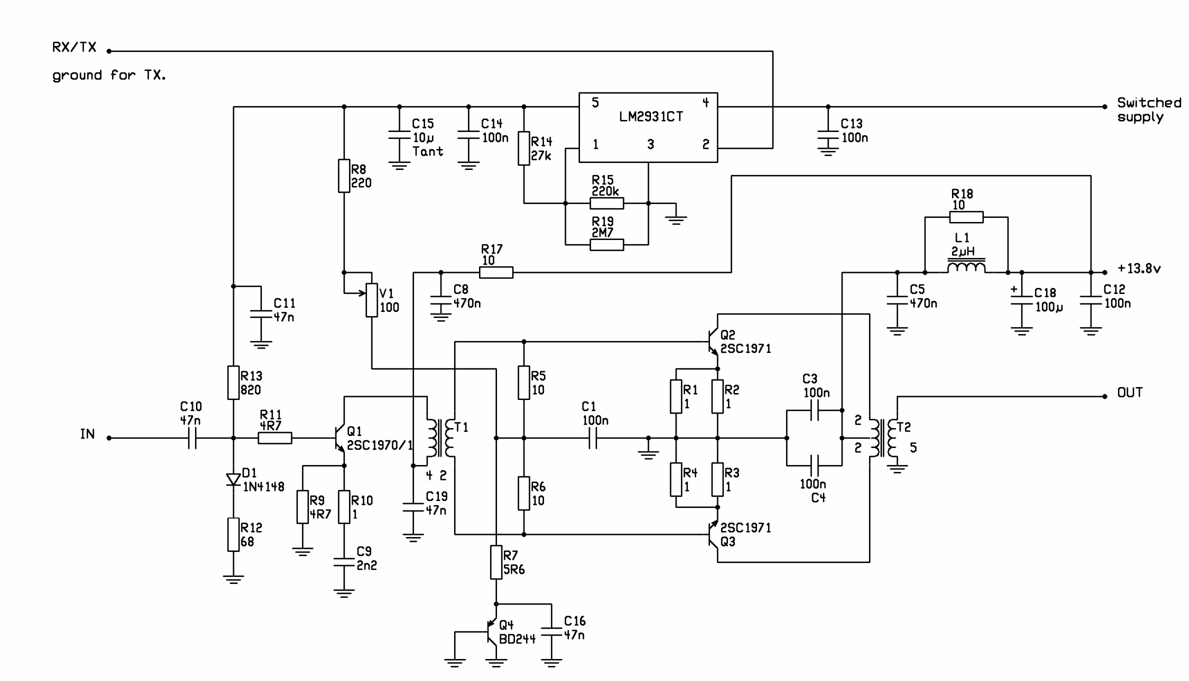

Fig. 25: PA Schematic

The main difference from the QRP-PA design is our use of a monolithic voltage regulator to provide the bias voltages and perform RX/TX switching. The component count is very similar, but the regulator gives better isolation from supply-rail fluctuations.

The topology is a pretty standard push-pull RF amplifier, and the symmetry helps to ensure a very clean output right up to the point where clipping sets in.

PA Construction and Testing

If you decide to buy the kit from Funkamateur, then follow the detailed construction notes which are supplied with the kit. Otherwise, construction follows similar lines to other QRP2004 boards, except that most of the components are surface-mount types, and all components apart from the power transistors mount on the track side of the PCB.

Since output power is only about 5W into 50Ω, you don't need a huge heatsink. If you intend to fit an aluminium back panel to your QRP2004, then as long as it is at least 1.5mm thick you could use that to dissipate the heat without the need for fins. Our first prototype used a small (recycled) heatsink 10cm x 7cm x 1cm deep, which worked well.

The wide copper strips at the top and bottom of our PCB layout are there purely as a drilling template for the power transistor mounting holes. Drill all PCB holes, hold the PCB in place on your heat sink, drill through the holes in the copper strips into the heatsink, and then re-drill the heatsink holes to 3mm, to accept M3 mounting bolts. Once the heatsink has been drilled, cut the 'template' strips off of the PA PCB before mounting any components.

Fit the links between PCB tracks and earth plane first, and then populate the rest of the board (apart from the four transistors), as shown by the component overlay diagram and component list in on our web site. There is only one wire link on this board – again this is apparent from the component overlay diagram. The LM2931 is mounted horizontally on the PCB over the lower centre copper rectangle.

L1 consists of 2 turns of 30 SWG enamel-coated wire on a ferrite bead. T1 and T2 are also wound with 30 SWG enamel-coated wire, but on Amidon BN-43-2402 'pig-nose' ferrite cores. T1 has two turns on the primary and four turns on the secondary. T2 has a four turn centre-tapped primary (i.e. two turns plus two turns) and a five turn secondary. Make sure that in each case the ends of the primary winding emerge from the core at the opposite end to the secondary winding. When you have finished, they should look like Fig. 26 and Fig 27.

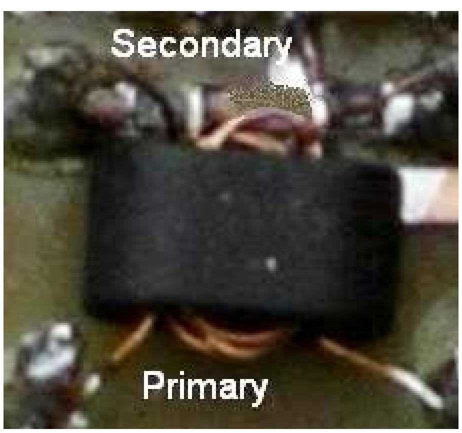

Fig. 26: T1 prototype

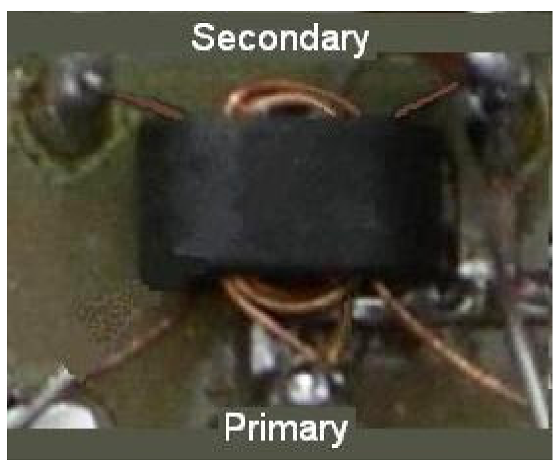

Fig. 27: T2 prototype

Copyright © 2004 - 2005 by QRP2004 Design Team

It is important to get the correct phasing of the windings when you mount T1 or you will probably find you have a 5W RF oscillator. The phasing of T2 is less important. When it's all put together your board should look a bit like Fig. 28.

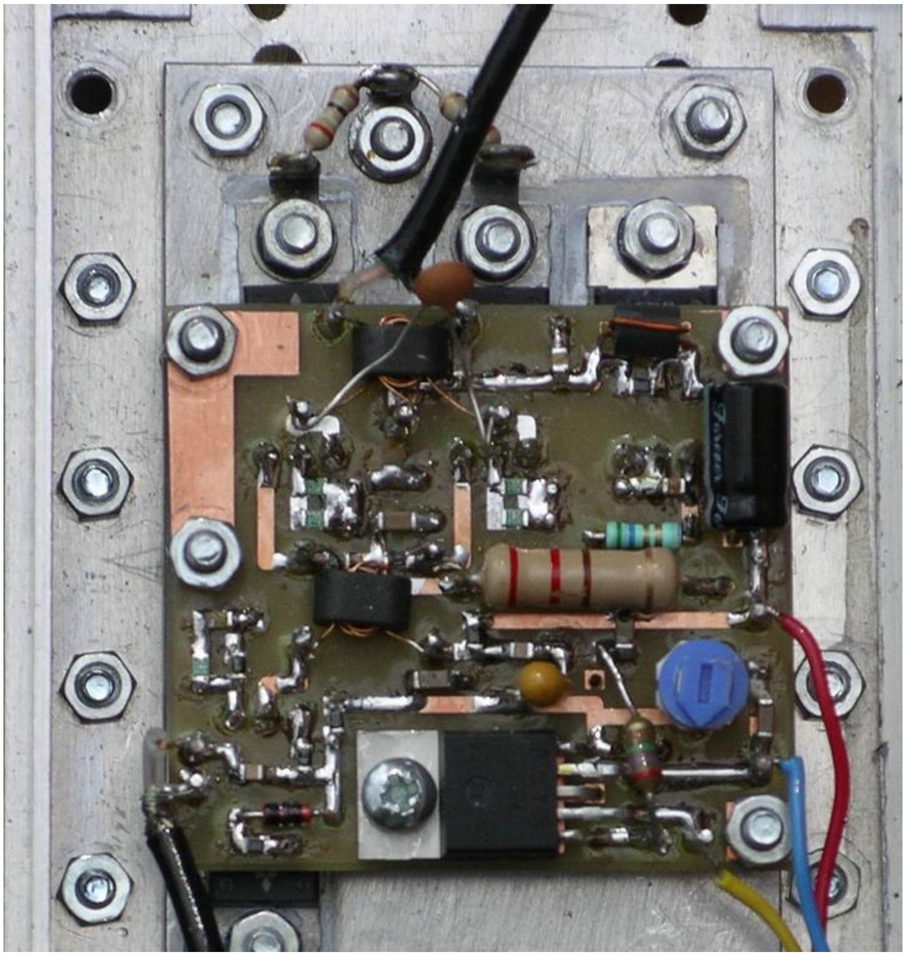

Fig. 28: Completed PA board

The PA board is actually supported at three corners by the power transistors. As you can see from the picture, the leads of the transistors need to be bent upwards at 90 degrees before soldering. For mechanical strength, the fourth corner is secured to the heatsink with an M3 bolt and short standoff pillar.

We found the easiest way to complete the PA assembly is to bolt the four power transistors to the heatsink (just finger-tight is fine for now) and then insert their leads through the appropriate holes in the PCB. You may need to adjust the leads a bit (and perhaps slightly rotate the transistors) to make everything fit nicely. The PCB should be as close to the heatsink as possible (to minimise lead lengths) whilst leaving some clearance to avoid short-circuits. Then solder the transistor leads whilst holding the PCB parallel to the heatsink.

The power transistors need to be insulated from the heatsink, so when they have been soldered in, remove the bolts again. Then remount the transistors using standard TO220 insulated mounting kits. As always, make sure that the heatsink is flat and the mounting holes are carefully de-burred before mounting the transistors.

During testing we found that some PAs were prone to parasitic oscillation. Peter Zenker kindly came to the rescue again, and suggested that the tab of each output transistor (Q2 and Q3) should be connected to the metalwork of the heatsink via a 39R resistor. You can see these resistors near the top of Fig. 28. The resistors help to damp a resonant circuit which appears to relate to the bond-out wires within the transistors. They may not be needed in your version of the PA, but we strongly recommend that you fit them anyway as a precaution.

Finally, solder 30cm lengths of thin coax to the input and output connection points of the PA board: we used RG174.

After the customary visual check, connect a 50Ω dummy load (at least 10W rating) to the PA output, turn preset V1 fully clockwise (maximum resistance), and apply 12V from a current-limited power supply to the board. At this point very little current should be drawn – at most 1 or 2mA.

Connect the RX/TX control line to ground, and the current should increase to about 100mA. If the current is much different, then you may need to adjust the value of R13 slightly. Then adjust V1 until the total current increases by a further 150mA. As you adjust V1 the current should increase smoothly: any sudden jumps in current suggest parasitic oscillation.

If all is well, the heatsink can now be attached to the outside of the back panel of the chassis via standoff pillars.

If you have an RF signal generator, it's worth checking PA performance in a bit more detail before wiring it up to the rest of the radio. Either monitor the output across the dummy load with an oscilloscope, or connect the dummy load via a power meter. Inject about 1mW (0dBm) at 1.8MHz into the input coax, and check the output power. Repeat for other amateur band frequencies. A prototype produced the results in Table 4: as you can see, PA gain varies by only about 3dB up to 30MHz, but tails off significantly by 50MHz.

Frequency	DC Current In	RF Power out
1.8MHz	0.9A	4.8W
3.5MHz	1.0A	5.3W
7.0MHz	1.0A	5.3W
10.1MHz	1.0A	5.3W
14.0MHz	1.0A	5.1V
18.1MHz	1.0A	4.8W
21.0MHz	0.9A	4.2W
24.9MHz	0.75A	3.1W
28.0MHz	0.65A	2.4W
50.0MHz	0.3A	0.25W

Table 4: PA output power for 1mW RF input

When you are satisfied that the PA board is working correctly, its RF input coax cable needs to be wired to the J5 (TXout) connector of the BPF board. The PA board output coax and the RX/TX control line are both waiting to be connected to the Low Pass Filter board, which we'll build next.

Low Pass Filters

The low pass filter (LPF) board is the most conventional part of the whole design. In keeping with many other HF all-band rigs we use six separate filters on the LPF board. The lower three HF bands each get a dedicated filter, but the higher bands 'double up' with each filter handling two bands. And of course the board also supports an optional seventh filter, for the 6m band.

In keeping with the BPF board, all signals apart from RF in and RF out are provided by the 16-way System Bus ribbon cable. Not surprisingly, the schematic in Fig 29 shows that this board is dominated by the filter sections.

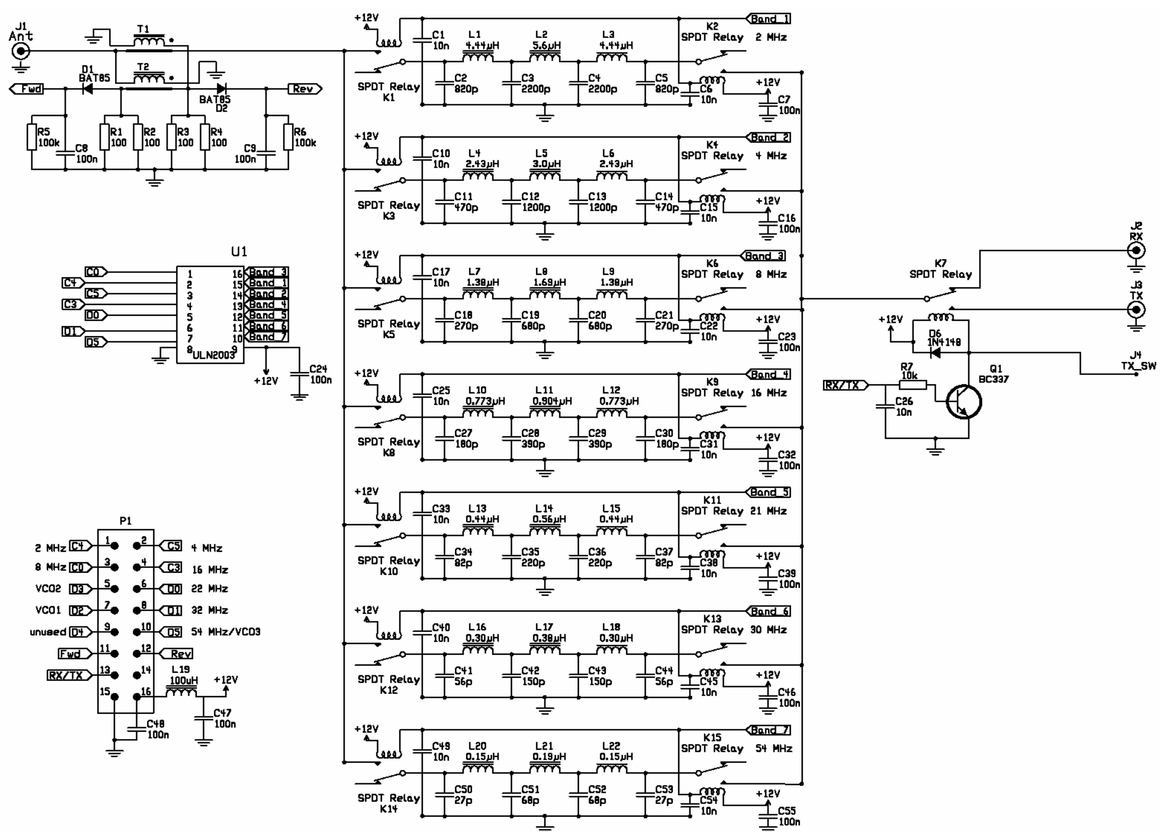

Fig. 29: LPF schematic

The filters are all 7-element Chebyshev designs, with 50Ω input and output impedances. Again, the component values are taken directly from the fantastically useful tables towards the back of the ARRL Handbook.

Since these filters have to withstand the mighty 5W RF output from our PA stage, we have shied away from solid-state switching on the LPF board. Instead, each filter is selected by a pair of miniature 12V relays. The relays are driven by a ULN2003, which boosts the low-power 5V band select signals from the control board as necessary. One further relay is provided to perform RX/TX switching, since the low-pass filters are used during both receive and transmit.

The remaining few components on this board constitute a 'Stockton' power bridge, which gives us an indication of forward and reflected power for use by the ALC circuit. (ALC is handled on the Band Pass Filter (BPF) board – see the BPF section for more details.) The authors are big fans of the Stockton bridge, preferring it in almost all cases to the more common Bruene bridge.

LPF Construction and Testing

Component layout on the LPF board is pretty compact, so take care over construction and build one filter at a time. As with the BPF board, use small ceramic disk capacitors for all of the filter sections. If possible, hand select capacitors which are close to their nominal value.

All of the filter coils are wound with 24 SWG enamel-coated wire on small Amidon toroids, and the table 5 shows the core type and number of turns for each coil:

Band	Coil	Type	Turns	Coil	Type	Turns
160m	L1, L3	T50-2	30	L2	T50-2	34
80m	L4, L6	T37-2	24	L5	T37-2	26
40m	L7, L9	T37-6	21	L8	T37-6	24
30m & 20m	L10, L12	T37-6	16	L11	T37-6	17
17m & 15m	L13, L15	T37-6	12	L14	T37-6	14
12m & 10m	L16, L18	T37-6	10	L17	T37-6	11
6m (option)	L20, L22	T37-12	9	L21	T37-12	11

Table 5: LPF coil winding details

In all cases the coil windings should be distributed evenly around approximately 80% of the circumference of the toroid.

T1 and T2 in the bridge circuit are wound on ferrite FT37-43 toroids. One winding consists of 12 turns of 24 SWG enamelled wire wound on the toroid, whilst the other winding is a single loop of plastic-insulated hook-up wire which passes through the centre of the toroid.

When you put it all together, your LPF board should look rather like Fig. 30.

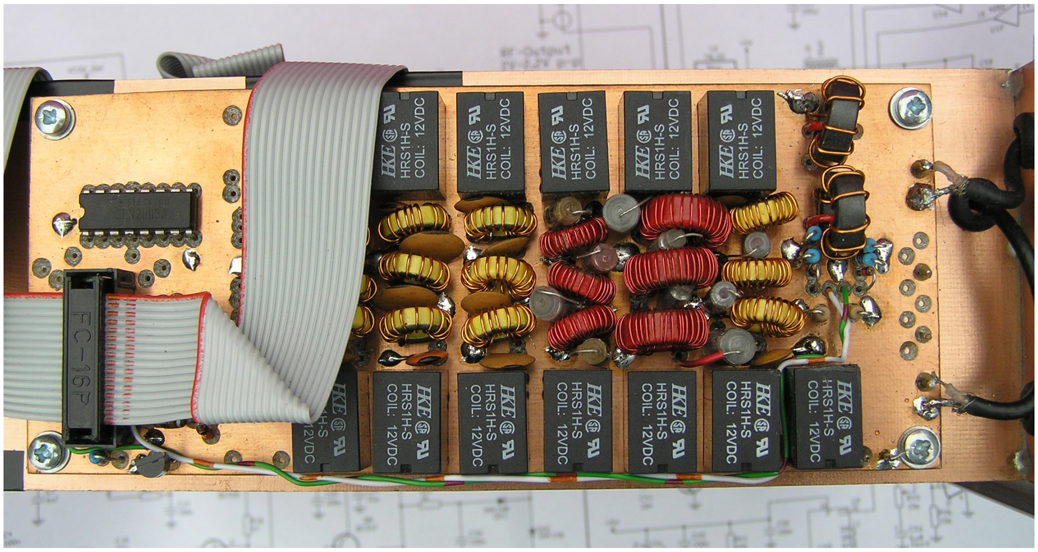

Fig. 30: Completed LPF board

After the usual visual checks, it's time to try it out. The other end of the 'System Bus' cable which you started to make during BPF board commissioning will need an IDC connector to be crimped onto it. As before, it's easier to do this when the LPF board has been mounted on the outside of the right-hand face of the chassis, again using M3 bolts and short metal spacers.

Dress the free end of the System Bus ribbon cable towards the LPF connector, so that the stripe on the cable coincides with pin 1 of the connector. When you are confident about the cable routing crimp on the IDC socket and plug it into the LPF board. Temporarily unplug the BPF board to limit damage in the event of an LPF problem.

Apply 12V to the control board from a current-limited PSU. Although current consumption depends on which make of relay you use, a current limit of 100mA should generally be sufficient. As power is applied, you should hear one pair of relays activate. By selecting different frequencies/bands on the control board, you should hear different pairs of relays being activated and deactivated. If you put the radio into transmit mode, you should also hear the RX/TX relay on the BPF board switch in.

There are no conventional preset adjustments needed (or offered) on the LPF board. However, as with the BPF board, we found that some of the coils may need adjustment to get the desired filter cut-off frequency. It's therefore a big benefit if you can check the response of each filter before moving on to final testing. Refer to the BPF board description for a rough-and-ready method for checking filter responses, but remember that whilst the BPF board has a single 16 – 32MHz filter, this range is covered by separate 16 – 24MHz and 24 – 32MHz filters on the LPF board.

Inject the RF signal into the J1 (Antenna) coax connection point and monitor the RF voltage at the J2 (RX) connection point. The measured cut-off frequencies from one of our prototypes are shown in table 6.

Displayed Frequency	Nominal frequency response	-6dB point
1MHz	0 – 2MHz	2.9MHz
3MHz	2 – 4MHz	5.1MHz
6MHz	4 – 8MHz	9.3MHz
12MHz	8 – 16MHz	16.6MHz
20MHz	16 – 24MHz	26.0MHz
28MHz	24 – 32MHz	38.3MHz

Table 6: LPF cut-off frequencies

You'll see that we've allowed a bit of extra 'lee way' on these filters to ensure that the in-band losses are very low. This is more important here than it is for the BPF board since any attenuation by the LPF board is seen as an impedance mismatch by the PA.

Once you are happy with your filter responses, connect J1 to your preferred style of antenna socket, J2 to connector J4 on the BPF board, and J3 to the output from the PA board (all using thin coax). Finally, connect the RX/TX Control Line of the PA board to the J4 (TX_SW) connection point on the LPF board, using insulated hook-up wire.

Final setup and testing

Final performance of the QRP2004 depends heavily on how well you have set it up, so it's worth taking some time over this. The ideal setup procedure requires access to a good quality RF signal generator and an oscilloscope with 50MHz or greater bandwidth. As a minimum you'll need an antenna, a DVM, a power meter which can accurately measure down to 5W of RF, and a dummy load.

Starting off with the good news, the mixer board requires no adjustments, and the LPF board should require no further adjustments (any coil tweaking should have been done at the time you built it). Similarly, the PA board bias control can probably be left alone, since that was set earlier.

That leaves the VFO, BPF, Audio and Control boards to set up.

VFO setup

Taking the VFO module first, the only adjustments needed are to the main VCO coils, to set the frequency range for each VCO. This should have been completed once already, but it's worth checking VCO alignment again (using the procedure outlined in the VFO description) now that all of the boards have been mounted on the chassis.

BPF setup (part 1)

Moving on to the BPF module, there are two preset potentiometers (R9 and R10) for the forward and reflected power signals which feed the ALC circuit. We'll come back to those again at the end, but for now both of these controls should be set for maximum output power. This is easily done by connecting the QRP2004 to a 50Ω dummy load via a power meter.

Copyright © 2004 - 2005 by QRP2004 Design Team

Hold down the CW key to produce a constant RF output and watch the output power as you adjust preset R9. At one end of its range R9 should reduce the output power significantly, so then you just turn R9 to the opposite end of its travel. R10 should have no noticeable effect, since there should be negligible reflected power from your dummy load. For now, set it to the same position as R9.

Audio board setup

By far the most time-consuming board to set up is the Audio board. There are a large number of presets, which are critical for good performance. We'll start off by optimising the received audio path, and then move on to the transmit circuits.

Presets R31 and R37 govern the rejection of the 'unwanted' sideband. They are most easily adjusted by injecting an RF signal from a signal generator into the antenna socket, but you can use the carrier from a strong broadcast station to achieve almost the same result. Either set your signal generator to about 1mW output at a frequency in the 40m band and tune the QRP2004 to that frequency, or connect an antenna and tune to one of the strong broadcast signals just above the 40m band.

Set the mode to lower sideband, and as you tune either side of the carrier you should hear the wanted signal as you tune above the carrier frequency and the unwanted signal as you tune below it. Tune down to the unwanted signal, and then adjust R31 and R37 to minimise the strength of the unwanted signal. You'll find that you need to have both of these presets close to the correct position before you get a significant null on the unwanted signal. This means that you need to search around for the right combination of settings. However, once you start to find the null it is easy to refine it by slight adjustment of each preset in turn, until the unwanted signal is almost completely cancelled.

Next set up the AGC, using presets R143 and R145. Audio-derived AGC is always a bit of a compromise, but we have found this circuit works quite well as long as it is set up correctly. To do this you'll need some way of measuring the AC signal voltage coming out of U12b (either an oscilloscope or a DVM with an AC Volts range that is accurate up to at least 1kHz). You'll also need a signal source - either:
a) an RF signal generator connected to the antenna input (somewhere around an S9 signal is ideal, corresponding to 50uV RMS), or
b) an AF signal source connected before U12b on the receive audio path (for example, at the point labelled RX_AF on the schematic), or
c) connect an antenna and tune to a strong stable signal, such as the carrier of a local AM broadcast station. The signal must be strong enough to ensure that the AGC is operating. Again, an S9 signal is ideal but don't rely on the QRP2004's S-meter because this won't work until the AGC has been set correctly!

The adjustment sequence is:

1. Turn R145 (22K) fully anti-clockwise, so that the slider is at minimum voltage.
2. Connect your AC voltage measuring device to pin 7 of U12, and activate your strong signal source. Aim for an audio frequency of roughly 1kHz.

3. Adjust R143 so that the output at pin 7 of U12 is 250mV RMS (about 700mV peak to peak).
4. Disconnect the signal generator or antenna, and wait until the background noise level rises to its maximum level (i.e. until the AGC gain has reverted to maximum).
5. Slowly turn R145 clockwise until the output level just starts to drop. This needs to be done very slowly because the AGC circuit is slow to react to R145.
6. Connect an antenna, tune to a strong CW or SSB station (CW is preferred) and wait until the station starts to transmit at the start of the next over. If the AGC audibly over-reacts, advance R145 very slightly more (i.e. turn it slightly further clockwise) and repeat this step.

The received audio path is now complete, so you can move on to the transmit path.

Connect your microphone and CW key to the radio, and connect a dummy load via an RF power meter to the antenna socket. (The power meter isn't used just yet, but you'll need it soon). Turn the front panel 'TX Drive' control to maximum, and set the frequency to somewhere in the 40m band.

The CW amplitude is set by preset R57. Use an oscilloscope to monitor the output of op-amp U1b (U1, pin 7) as you key the radio. Use R57 to increase the CW oscillator amplitude until you can see the sinewaves start to be 'rounded off' by the compression circuit. Then back R57 off slightly to keep the sinewaves undistorted. If you don't have an oscilloscope you could use a DVM on its AC Voltage range, and set R57 to give about 0.15V RMS on the output of U1b. While you are in that area, connect a loudspeaker to the QRP2004 and adjust preset R59 for your preferred sidetone volume.

Locate preset R5 on the audio board. Key the radio again and adjust R5 to give about 7W of RF output from the radio.

Preset R8 determines the CW 'hang time' – the delay before the radio switches back to receive mode after releasing the CW key. The relay switching in the QRP2004 is not suited to QSK keying, so you should adjust R8 so that the QRP2004 does not drop out of transmit mode during normal keying. This will depend on your keying speed, so you'll need to experiment a little to get it right.

The next step is to optimise suppression of the unwanted sideband when transmitting. This requires correct adjustment of the three presets on the output of the polyphase network – R101, R104 and R105. You can set these presets much more accurately if you have access to an oscilloscope. With a dummy load still connected to the antenna socket, set the frequency to somewhere in the 40m band and hold down the Morse key again. Set the oscilloscope for a signal amplitude of about 50V peak to peak, and connect the probe across the dummy load. Adjust as necessary so that you can clearly see the envelope of the output signal.

The upper and lower edges of the envelope will appear wavy, and you need to adjust R101, R104 and R105 to make the envelope as smooth as possible. In practice you won't achieve a completely flat and uniform output signal envelope since our CW oscillator has some low-

Copyright © 2004 - 2005 by QRP2004 Design Team

level harmonics on it, but you certainly should get close. Be warned that it isn't easy to get all three presets adjusted correctly, so take your time and be persistent!

If you don't have the use of an oscilloscope, connect a high impedance DVM, on its AC Voltage range, to each of the TX_0, TX_90, TX-180, and TX_270 outputs in turn. Use the presets to match the amplitude of the other three outputs to that of the TX_0 signal.

Having set up the audio board for CW operation, we'll move on to SSB mode. The only preset which is specific to speech operation is R16, the microphone compression preset. However, this can be difficult to adjust correctly due to the high peak-to-average ratio on speech waveforms, and the limited compression which we apply. (If you prefer to operate with higher speech compression levels then you could use a microphone with built-in compression.) If you have an oscilloscope, monitor the output of U1b again. This time, select LSB mode and speak into the microphone. Adjust R16 (compression preset) until you can see speech peaks being gently 'rounded off' by the compression circuit during normal speech. This should happen at around 0.6V peak-to-peak output from U1b. If you don't have an oscilloscope, increase the microphone gain with R16 until you get a peak SSB output of about 7W.

BPF setup (part 2)

Now we need to revisit the ALC controls on the BPF board. Hopefully your QRP2004 is still connected to a dummy load via a power meter. Select a frequency in the 40m band, and hold down the CW key. Adjust preset R9 on the BPF board (forward power control) to set the output power to about 5W. At this point the other preset (R10) should still be at one extreme of its travel. We recommend that you simply turn this back to the opposite extreme to give the maximum protection against damage caused by high SWR. Check the power output into the dummy load on each amateur band to confirm that the ALC is working effectively, so that you see 5 to 6W out on all bands apart from 6m.

Control board adjustments

The only physical adjustment on the control board is the preset for LCD display contrast, which is easily set by eye.

There are also two software-controlled adjustments to complete. The first of these sets the 'zero level' for the S meter. Assuming that you are still connected to a dummy load, just press '*' and then '0' on the keypad to invoke function 0. This reads the current AGC voltage and remembers it as the 'no signal' level.

The final adjustment provides accurate calibration of the DDS reference frequency. Our DDS is clocked from a standard crystal clock module, and whilst this should be sufficiently accurate in frequency for most purposes, we offer the option of calibrating the software to improve the accuracy of the displayed frequency. Follow these steps to calibrate the DDS reference frequency:

1. Make sure RIT is disabled, and select USB mode.

2. Tune to where a strong broadcast station should be, for example 6195kHz for the BBC World Service. Do NOT tune until the signal sounds right, tune to exactly 6195000Hz (for example) on the display. (Note that stations at higher frequencies allow more accurate calibration.)
3. Enter * 1 on the keypad to enable RIT.
4. Use the RIT control to tune the signal until it sounds exactly right.
5. Press '#' to accept the RIT setting, then enter * 2 to perform the calibration.

After this sequence the necessary correction to the reference frequency will be calculated and stored, RIT disabled, and the radio re-tuned, based on the new reference frequency. You should hear no significant change to the tuned signal.

Power cycle the radio, select USB mode and tune to an AM signal at a known frequency, tuning for zero-beat with the carrier. The display should now exactly match the AM carrier frequency.

Measured Results

The following performance measurements are based on two prototypes fully operational both on transmit and receive and a third prototype operational on receive only.

Sensitivity measurements were made with a recently calibrated RF Generator connected to the antenna input of the QRP2004 in SSB mode and with the SCAF filter at maximum bandwidth. For SINAD measurements we use DL4YHF's excellent Spectrum Lab software package connected to the headphone output of the QRP2004. All measurements were made at 1 kHz for 10dB SINAD with correction for standard CCITT weighted filtering applied.

RF Power measurements: These were taken in CW mode into a 50Ω dummy load whilst ensuring that the PA is not overdriven.

Band	RF Power Watts	Sensitivity uV for 10dB Sinad
160m	6.0	0.22
80m	5.7	0.25
40m	5.4	0.20
30m	5.8	0.20
20m	5.9	0.30
17m	6.0	0.25
15m	6.0	0.35
12m	5.8	0.30
10m	5.0	0.28

Table 7: RF Power and Sensitivity Measurements

Selectivity: 2.5 kHz at -6dB and 4kHz at -60dB (SCAF maximum BW).
Measurement was made before the AGC stage by connecting the input of a PC soundcard to RX_AF (C94) on the audio board, and adjusting RF Generator level and frequency.

Side Band Suppression: Over 45dB has been measured on frequencies below 10MHz reducing to -38dB measured on 28MHz.

RX 3rd Order Intercept point: It is difficult to make this measurement accurately, however we believe this will at least +20dBm based on results others have achieved with the Tayloe mixer.

RX Spurious Responses: although this is not an exhaustive list, the main spurious responses are:

1) the first sub-harmonic of the tuned frequency (e.g. unwanted reception of a strong signal at 3.5MHz signal when tuned to 7.0MHz). Rejection of the first sub-harmonic is about 55 – 60dB.
2) A sequence at 500kHz intervals, starting at 500kHz and decreasing in power with increasing frequency. These are assumed to be harmonics of one of our internal oscillators, but we have not localised the source yet.
3) Harmonics of the clock signal to the SCAF. These are usually close to the noise floor in level, but a few are stronger. If troublesome they can be easily worked around by changing the audio bandwidth setting.

RX Local Oscillator leakage: -65dBm at 7MHz, increasing to -53dBm at 28MHz. (For comparison purposes, ETSI limits for commercial equipment are -57dBm.)

TX harmonics: worst cases are -40dBc (3rd harmonics of 14MHz, 21MHz and 24.9MHz). Other harmonics better than 45dB below carrier.

Further practical information

To keep the length of a magazine article to a reasonable length, additional information like PCB lay-out and PCB component placement diagrams for the QRP2004 and other information will be made available in parallel through a dedicated QRP2004 Website: (Ref. 11) http://myweb.tiscali.co.uk/qrp2004/

We will try to answer or clarify questions as result of publication of this project, however please accept that we have only limited time available to reply to detailed correspondence.

- **Component Availability:** Most components are relatively standard and can be obtained from the following established component suppliers to UK Radio Amateurs. We use JAB Electronic Components, SYCOM or Mode Electronics (22-24 Warstone Lane Hockley, Birmingham B18 6JQ). Mode supplied us with the Philips 74HCT9046.

Copyright © 2004 - 2005 by QRP2004 Design Team

- Farnell is used for other ICs like SMD ICs or 74AC series as well as Bourn shaft encoder and 1% resistors.
- Keypad came from Maplin.
- Blank PCB sheet for scratch building chassis was acquired at various Amateur Radio rallies over the years.
- Membership of G-QRP club is recommended, MV209 varicap diodes were obtained from their club sales.

Future developments

Although we consider we have met our goals of the QRP2004 project as published in this article, we would not be radio amateurs without contemplating a number of potential performance improvements in the future: Some are obvious and quite easy to implement, however we decided we did not want any further delays in publishing this design.

- Improved Sensitivity: the 14dB TX Amplifier (Q2 BFR96) could be used as a switchable RX Amplifier as well. This is would only require an additional 2 pole relay, a simple firmware change and a PCB lay-out change. This would improve the usable sensitivity on the higher bands above 20 MHz albeit at reduced 3rd order IP performance. It is likely that this would improve the 6 meter sensitivity to an acceptable level, although there are other areas of 6m performance which we would still need to address.

- Digital Audio AGC derived from the PIC processor.

- DSP Processor replacing the audio board leading to improved performance: most functions on the Audio board could be replaced with a DSP board like the KDSP-10 kit from TAPR Group. See recent article in Radcom

- State of the Art Amateur Band only performance. The VFO module could de redesigned for narrow band WARC band only operation with VCOs operating at UHF. Bandpass filters made tunable for Amateur Band operation only (Cohn filters) to augment the improved VFO noise performance.

- Higher RF output power – external amplifier to increase RF power output from the current 5 Watt QRP level.

No doubt you will have your own ideas: we are keen to encourage further experimentation, but please share your work….

Credits

We would like to express our great gratitude to the following people for their ideas and encouragement:

- Dan Tayloe, N7VE, for his fine work on the Tayloe Product Detector, and for bringing the whole QSD concept to our attention,
- Gerald Youngblood, AC5OG, for revealing the final piece of the jigsaw while we were struggling to produce a clean RF output signal.
- Peter Zenker,: DL2FI, for his delightfully simple but effective 5W PA design,
- Klaas Spaargaren, PA0KSB (SK) for his low-noise VFO design, and for generally being such an inspiration to us over many years.
- Joris van Scheindelen PE1KTH for his simplifications and technical notes on PA0KSB's VFO design.
- Steve Hunt, G3TXQ, whose audio frequency CW Oscillator and keying circuit formed the basis for our variant, and
- Eep van Hardeveld for reviewing early drafts of this article, encouragement and very valuable feedback.

References

Ref 1: The QRP2000 Project Home Page
http://www.stevef.demon.co.uk/qrp2k/qrp2000home.htm
Ref 1a: FDIM Symposium Proceedings 1999 Dayton OH Sponsored by QRP-ARCI
Page 53-60

Ref 2: The QRP2001: An Experimental Receiver
Journal of the G-QRP Club - Sprat 101 Winter 1999/2000 Page 19 - 23

Ref 3: The QRP 2001 Project Home Page
http://www.qrp2001.freeserve.co.uk

Ref 4: US patent office – Patent No 6230,000 Tayloe Oct 15 1998. This can be found on the internet at http://patft.uspto.gov/netacgi/nph-Parser?Sect1=PTO1&Sect2=HITOFF&d=PALL&p=1&u=%2Fnetahtml%2FPTO%2Fsrchnum.htm&r=1&f=G&l=50&s1=6230000.PN.&OS=PN/6230000&RS=PN/6230000
You may need to download a suitable image viewer, such as the Apple QuickTime™ plug-in, to see the images. A free version of QuickTime is available for download from http://www.apple.com/quicktime/download

Ref 5: Technische Notities by PAOKSB: Klaas van Spaargaren PAOKSB (SK)
Electron Journal of the Dutch Amateur Radio Society "VERON" March 1999 and June 2000
Ref 5a: Ervaring met de DDS-synthesizer van PAOKSB: Joris van Scheindelen PE1KTH
Electron Journal of the Dutch Amateur Radio Society "VERON" August and September 2000

Ref 6: Software Defined Radio for the Masses, Part 1 to 4: Gerald Youngblood AC5OG QEX Jul/Aug 2002 - Mar/Apr 2003. Part 4 is particular relevant: see http://www.flex-radio.com/articles_files/SDRFMP4.pdf

Ref 7: Polyphase Network Calculation using Vector Analysis Method: Tetsuo Yoshida JA1KO, QEX June 1995 Page 9-15
Ref 7a: Comparison of SSB phasing methods: Lawrence D Woolf GJ3RAX, Radio Communications April 2004, page 68 and 70

Ref 8: Excel Modelling Macro by James Verduyn
http://www.qrp2001.freeserve.co.uk/polyphase.xls

Ref 9: Steve Hunt G3TXQ Transceiver, Radio Communication, February 1989 page 45

Ref 10: DL4YHF's Software Archive: WinPic - A PIC Programmer for Windows
http://www.qsl.net/dl4yhf/winpicpr.html

Ref 11: QRP2004 Web Site: http://myweb.tiscali.co.uk/qrp2004/

Copyright © 2004 - 2005 by QRP2004 Design Team