The Tucker Tin Mark 2*

An SSB Transmitter of simple design

Part 2

In this second article describing his easily-constructed solid state SSB transmitter, the author discusses the remaining circuit sections and such constructional matters as home etching of the printed wiring boards. If your interest was aroused by the first article, this one should start you reaching for the soldering iron.

by FRED JOHNSON, ZL2AMJ

Head, Department of Electronic Engineering
Central Institute of Technology, Petone, New Zealand

The remaining printed wiring board to be described is the VFO-mixer board. As shown in the block diagram given in figure 1 last month, this module accepts the 9.0MHz SSB signal from the balanced modulator board (leads G and H) and converts it to some frequency in the range 3.5 — 3.9MHz. The 80 metre signal emerges from lead K.

Figure 8 shows the detailed circuit. Q7 is the variable-frequency oscillator. This is a conventional Clapp circuit, using a field-effect transistor. It tunes 5.0 to 5.5 MHz. The components C28 and L6 are adjusted during setting-up for this frequency coverage. The output from the oscillator is taken via C32 to the base of Q8. This is an emitter-follower stage, its purpose being to isolate the VFO from the mixer.

Q9 is the mixer. This has two inputs. R24 is an emitter load which is common to both Q8 and Q9. The VFO signal is therefore injected into the emitter of Q9. The 9 MHz SSB signal is fed to the base of Q9 via L8 and its associated tuning capacitors C37 and C38.

The output from Q9 is taken from the collector. The output frequency is somewhere between 9.0 minus 5.0 MHz (i.e., 4.0 MHz) and 9.0 minus 5.5 MHz (i.e., 3.5 MHz), depending upon the actual VFO frequency setting. The collector tuned circuit must therefore be arranged to tune 3.5 to 3.8 MHz (the 80 metre band). I have stated that the VFO tunes 5.0 to 5.5 MHz. By changing L7 and C35 to another value it is possible to take an output between 9.0 plus 5.0 MHz (i.e., 14 MHz) and 9.0 plus 5.5 MHz (i.e., 14.5 MHz) from this point. The 20 metre band extends from 14.0 to 14.35 MHz so both 80 and 20 metre outputs are possible from Q9 with this 5.0 to 5.5 MHz VFO.

The slug in L7 is the "mixer tune" control and is made separately adjustable, as the "mixer peak" control, but more of this later when the construction of the rig is discussed.

The output to the Power Amplifier (PA) grid is taken from the collector of Q9. A signal from the anode of the PA stage is fed back to the other end of L7 (lead N). This is for neutralising the PA stage against self-oscillation, and will also be discussed later. Lead N was not shown on the block diagram of figure 1.

The action of D5 in the emitters of Q8 and Q9 will be mentioned later.

The mixer and VFO board layout is given in figure 9. Construction is the same as for the previous boards. Capacitor C36 is earthed to the chassis at the same point where R27 (PA valve grid leak) and C43 (cathode decoupling capacitor) are earthed.

The power amplifier stage uses the transmitter's only valve. The circuit diagram for this stage is shown in figure 10. Its function is to accept the signal from the output of the mixer Q9 and amplify it before passing it to the antenna.

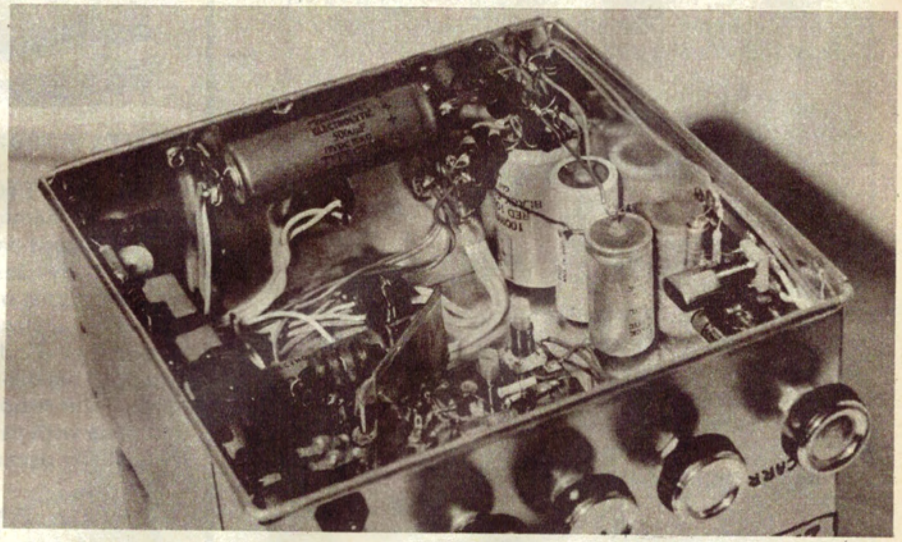
The circuitry for this stage is wired using tagstrips where required. As shown in figure 10, some of the wiring is below the chassis and some above. Note that the +300 HT does not appear above chassis—ie. C42 is kept below chassis.

A shield is mounted across the valve socket and is soldered to the chassis on each side of the socket, and to the centre spigot of the socket itself. This separates pins 1, 2, 3 and 9 from pins 4, 5, 6, 7 and 8. Figure 11 gives the construction details.

The test point in the cathode of the valve enables the cathode-current to be measured easily (described later). The stage uses a type 12BY7 valve and delivers about 4 watts Peak Envelope Power (PEP) output into a 50 or 75 ohm load.

The voltage across the load is metered by a simple RF detector circuit D6 and M, with their associated components. The meter is used during alignment and during subsequent tuning-up activities.

The construction of the PA stage is quite straightforward and will be treated later. It could be changed to operate on the 20 metre volt band by changing L10, but this has not been tried. The Tucker Tin Mk. II is presented as an 80 metre rig but constructors are welcome to try it on 20 if they wish. It is suggested that you get it going on 80 before trying 20!

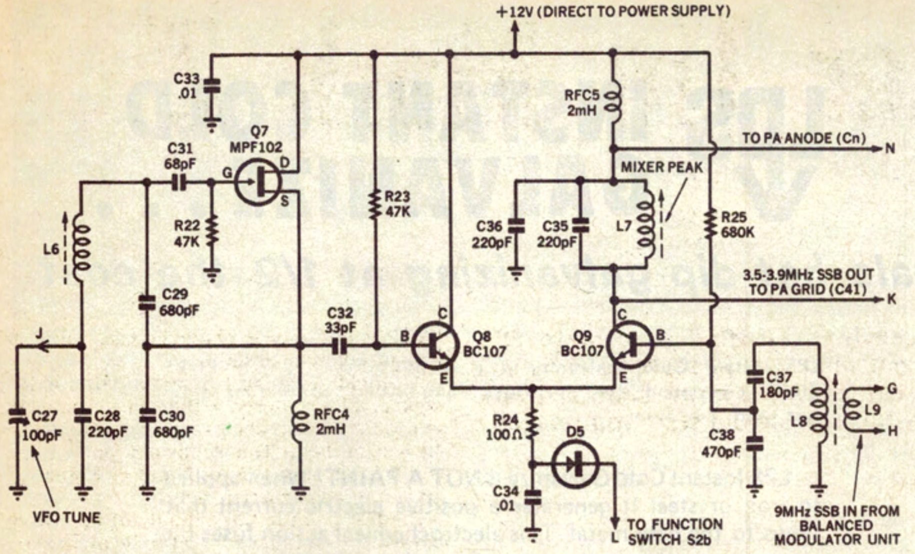


A view of the underside of the chassis, seen from the front

The output from this amplifier is capable of driving a subsequent "high power" linear amplifier to full amateur legal ratings. Such an amplifier, built on a matching chassis and panel, could be produced by you to your design quite easily. This is recommended as a later project.

The neutralising capacitor Cn consists of three turns of hook-up wire twisted tight around the lead to the RF choke RFC6. C47 and L11 form a 5.5 MHz VFO trap to remove

* This article is reprinted from the August 1971 issue of "Break-In", the official journal of the New Zealand Association of Radio Transmitters Inc., by arrangement.



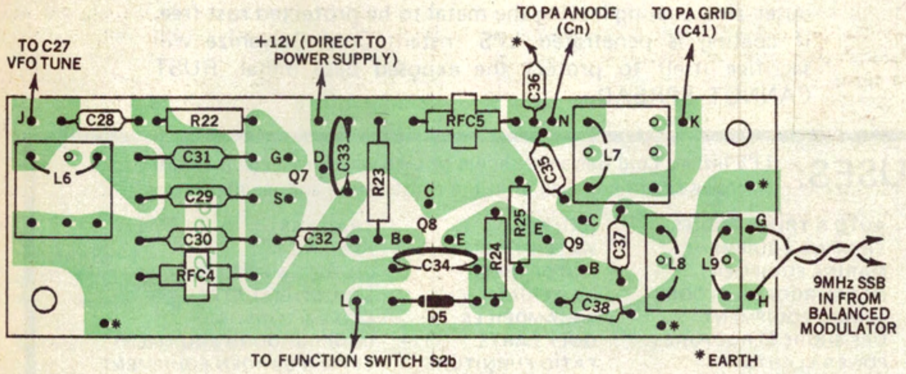


Figure 8 (top): The circuit of the VFO and mixer wiring board.

Figure 9 (above): The board pattern and component layout.

any direct VFO signal from the output. Details of adjustment are given later.

The power supply uses a conventional full-wave rectifier for the +300 volt supply and a voltage-doubler for the +12 volt supply. Fig. 12 shows the circuitry. This too is wired using tag-strips where necessary. Layout and other constructional detail is not critical.

The transformer used is an inexpensive type ideal for the task. It has two secondary windings, one is 250 volt aside at 42mA and the other is 6.3 volts at 2.3 Amps. Diodes D7 and D8 should be 1000 volt PIV (or greater) types rated at 500 mA or 1 Amp. Diodes D9 and D10 should be 50 volt PIV (or greater) types rated at 500 mA or 1 Amp. Because of the negligible price differences between these ratings, it is suggested that all diodes D7 to D10 inclusive could well be 1000 volt 1 amp types.

A mains on / off switch has not been fitted to the original rig. This could however be easily provided by using a potentiometer with a switch as the microphone gain control. The switch section should be placed in series with the mains active lead.

Fig 13 shows the connections between the various boards. The leads are lettered to permit cross-referencing with the circuit diagrams for each individual board. Note that the power supply and PA stage are not shown on this diagram. It will be noticed that many of the front panel components below chassis appear on this diagram — the microphone and key jacks, the audio gain control, sideband switch S1 and the function switch S2.

S1 is simply a change-over switch that swaps over the two audio leads between the audio board and the balanced modulator board. A double-pole changeover toggle switch could have been used. here.

S2 performs several functions. One pole (S2C) is completely isolated from the entire Tucker Tin Mk. II circuitry. It can be used to control the receiver to give send / receive operation of the complete station. Two wires are taken from S2C to the receiver. Possible methods for receiver connection will be mentioned later.

S2A applies the +12 volt supply to the audio preamplifier when SSB operation is required. This +12 volts is removed from the audio preamplifier when tune-up or netting functions are being carried out. This prevents noises inside the shack from modulating the rig. When "netting", +12 volts is applied via R31 to the balanced modulator to cause unbalance and hence a small radiated carrier for netting to the local receiver. The value of R31 is selected to produce a signal of sufficient output for the netting action. R32 creates a larger unbalance so that a carrier signal of much larger amplitude is radiated for tuning-up (this same signal is turned on and off by the CW key when CW operation is desired).

S2B controls the transmitter. The transmitter radiates a signal in SSB, net and tune positions, and is rendered inactive in the receive position (but all voltages are still applied). Both the VFO and 9MHz crystal oscillator run continuously on both transmit and receive. S2B is in series with the cathode of the 12BY7 so that this amplifier is rendered inactive when on receive. This eliminates a possible noise-radiation problem that was reported by several users of the original "Tucker Tin." S2B also switches off the mixer Q9 and the isolator Q8 by breaking their common emitter to earth lead. Because of the common DC connection between the cathode of the 12BY7 and the emitters of Q8 and Q9 when S2B is in the receive position, the voltages across the internal junctions of Q8 and Q9 can rise to high levels and cause damage to these two transistors. To prevent this, D5 (on the VFO and mixer board — figure 8) has been included to block this DC feedback voltage. D5 can be any diode with 50 volt PIV (or greater) and 100mA (or greater) rating. I suggest that it be the same type as diodes D7 to D10 (as used in the power supply) to simplify the ordering of parts.

A jack for the CW key enables the transmitter to be turned on and off when the key is operated. When the key is removed, the closed-circuit contacts operate to permit S2B to operate the transmitter for phone operation. When the key is plugged in the key must be held down to permit both netting and tuning — for CW the switch is left in the "tune" position.

All resistors shown on all diagrams are quarter-watt types unless otherwise stated. Half or one watt types are acceptable but may be bulky and difficult to mount in some positions. Similarly, all capacitors can be 12 volt working types unless stated otherwise. A complete list of all the components necessary was given in the first of these articles.

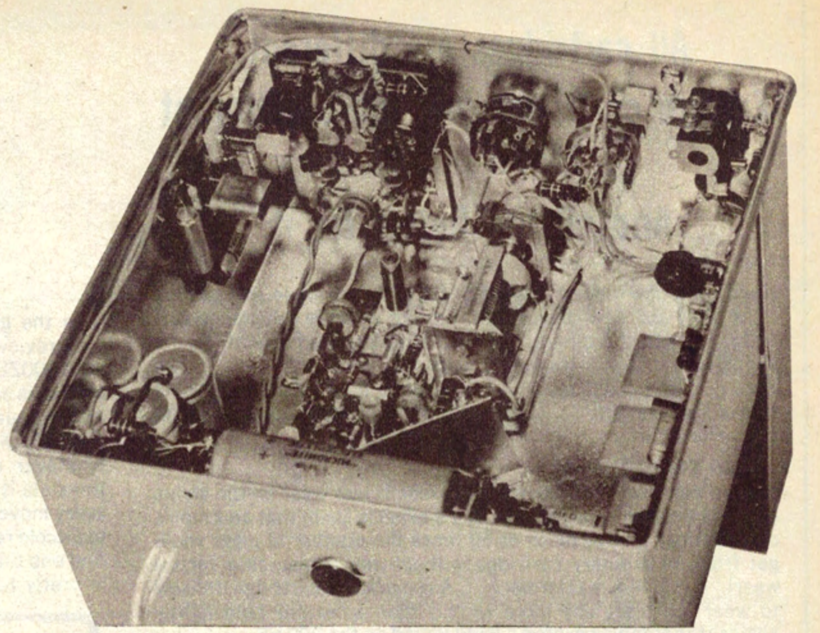
The transistor types quoted are the ones I used, but there are thousands of possible substitutes for each type. The connections for the specified types are shown in the small diagram, but I leave it to you to sort out your own substitution problem! Some component values may have to be changed when other transistor types are used (e.g. R7, R10, R23 and perhaps R25). These values may also have to be changed due to the spread of transistor parameters between individual devices of the same type. I used BC107B's but there is no reason why BC107A's or BC107C's should not work as effectively.

The layout diagrams given for each of the printed wiring boards differ a little from the prototype boards shown in the photographs. This is a direct result of developments during the production of the prototype. The holes visible on the rear apron of the chassis are likewise evidence of the power supply evolution!

The layout for each of the circuit boards has been given already and the problem now is to evolve a simple means for converting these into hardware. The following method is only one of many possible methods but is suggested to those who have not previously played with printed-circuits. Each step will be numbered so that some logical sequence can be followed. You will be surprised how easy they are to make.

1. Obtain the copper-clad printed-circuit

At right is another view of the underside of the chassis, as seen from the rear.



board stock and cut to the size quoted for each board. Either the "sharp hacksaw and cut" or the "scriber with rule then break" method can be used. File all edges clean and square.

2. Polish the copper surface clean, using steel wool or a kitchen abrasive cleaner. Keep your fingers off the freshly polished copper — handle it by the edges.
3. Place the board behind the layout diagram, copper side up. Use carbon paper and a ball point pen and trace the shaded area outlines on to the copper (i.e. ignore the components). The shaded areas represent the conducting areas of copper that are to be retained. Use new carbon paper.
4. You again have a choice here. You can use either a marker pen with ink that is not soluble in water, nail polish, or paint lacquer. The aim is to paint over THE COPPER AREAS TO BE RETAINED. All the areas to be retained must be covered, leaving the unwanted copper areas exposed. This work must be done carefully and with clean "edges" to ensure a craftsman-like result. Let the boards dry until this "resist" is thoroughly dry.
5. The exposed copper areas are now to be etched away. Ferric chloride (obtainable from chemical suppliers) is perhaps the best etchant to use. Use a plastic or glass dish, big enough to take the largest board. Fill with hot water from the kitchen tap. Sufficient liquid is required to cover the board. Add the ferric chloride, three or four heaped teaspoonfuls to each cup of water. Use a plastic spoon as a stirrer. Ensure that all the powder and lumps dissolve.
6. Insert the board, copper side up, making sure that it is completely immersed. Frequent agitation is desirable. The etching can take any time from 10 minutes to 2 hours depending upon the temperature and strength of the solution and the amount of copper to be removed. Inspect it frequently to see progress.
7. When all traces of exposed copper have disappeared, remove the board, wash in cold running water, and dry thoroughly.
8. Using methylated spirits or some other solvent, remove the resist to expose the copper conducting areas.
9. Dry and repolish if necessary.

The board should now show areas of shiny copper, exactly the same as the layout diagram. All boards are treated the same way, and can be made singly or together as you wish.

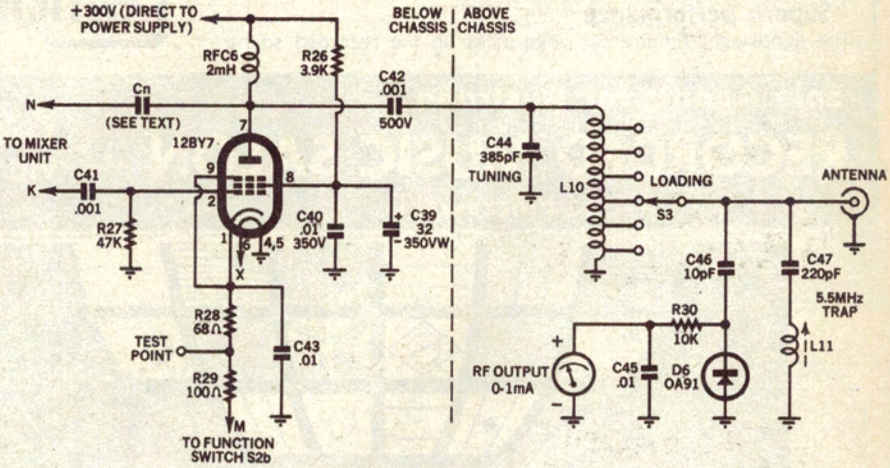


Figure 10: The circuit of the PA stage, which uses the sole valve employed in the transmitter. RF output is 4 watts PEP.

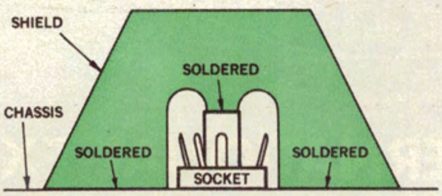


Figure 11: The construction of the shield fitted to the PA valve socket.

Some practice with some scraps of printed circuit board is recommended before you try the actual boards themselves!

Printed wiring boards usually have lots of holes so that the component leads can feed through the holes to the copper side where the leads are then soldered to the copper. This form of construction has all the components on one side of the board and the copper on the other side and requires the board to be mounted on spacers to keep the copper side away from the chassis. Considerable simplification is possible by soldering the components direct on to the copper side of the board. This eliminates the hole-drilling task. It also simplifies construction because you do not have to keep turning the board over all the time to check connections! The board can also be mounted flat on the chassis thus avoiding spacers (see figure 14).

This form of construction has been adopted in the "Tucker Tin Mk. II" and really makes wiring up very straightforward. The components are soldered on to each board where shown on the layout

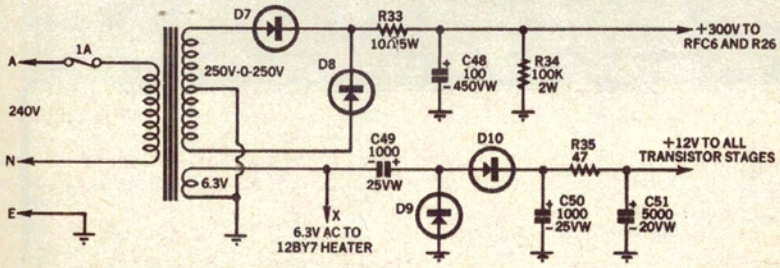


Figure 12: The circuit of the power supply used in the transmitter. For simplicity all diodes may be 1000V 1A devices if desired.

diagrams. Each board can be checked against the layout diagram when completed. Beware of short-circuits. Make sure that each component makes contact only where shown (unless the component is insulated). Capacitors, resistors, coils, diodes, transistors, chokes, all are wired direct on to the board. Even the connecting wires to other boards and switches (etc.) are soldered direct on to the copper where shown.

Use a 25 watt soldering iron, and a good quality multicore solder.

The photographs show the general layout of the chassis and the positioning of the boards. The basic chassis is a cake tin 8 inches by 8 inches by 4 inches deep. The front panel is another cake tin, 8 inches by 8 inches by 2 inches deep. These two items are obtainable from chain stores. It is recommended that these tins be used as chassis and panel because they are very easily soldered. The front panel itself is soldered to the chassis. Even the valve socket, tag strips, and the print-boards themselves are soldered into place. This saves a lot of drilling and messing about with nuts and bolts that would be necessary if any other type of chassis was used. In fact, nuts and bolts are only required to

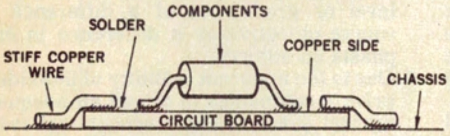


Figure 14: Component mounting and board mounting may be performed in this way.

mount the power transformer, meter, final tuning capacitor and coil, and are not required for any other task.

Very few holes have to be drilled — and most of these are for mounting the controls on the front panel.

One side of the smaller cake tin is cut out and this discarded side can be used to fabricate the shield across the valve socket and the VFO capacitor mounting bracket. Both cake tins are soldered together to form the chassis and panel arrangement as shown in the photographs.

The balanced modulator board is mounted by pushing the threaded mounting sections of the carrier balance potentiometers through holes in the chassis and front panel and bolting on with additional nuts. It is suggested that this board be mounted first. It still requires earth straps to "earth" the copper earth sections on the board to the chassis.

The VFO tuning capacitor (C27) is mounted on a small bracket made from another scrap of tin. This bracket is soldered in position. A Jackson Brothers planetary reduction drive is used as the dial mechanism. The VFO capacitor mounting must be rigid otherwise drift and frequency instability problems could occur. The bracket should be braced to the front panel with stiff wire straps.

The front panel switches (S1 and S2) and audio gain control should be mounted next.

The components above the chassis should next be fitted — meter, tuning capacitor, loading switch (S3), and final tank coil L10. Once these are positioned the VFO and mixer board can be mounted. This enables the valve sockct to be positioned. The shield

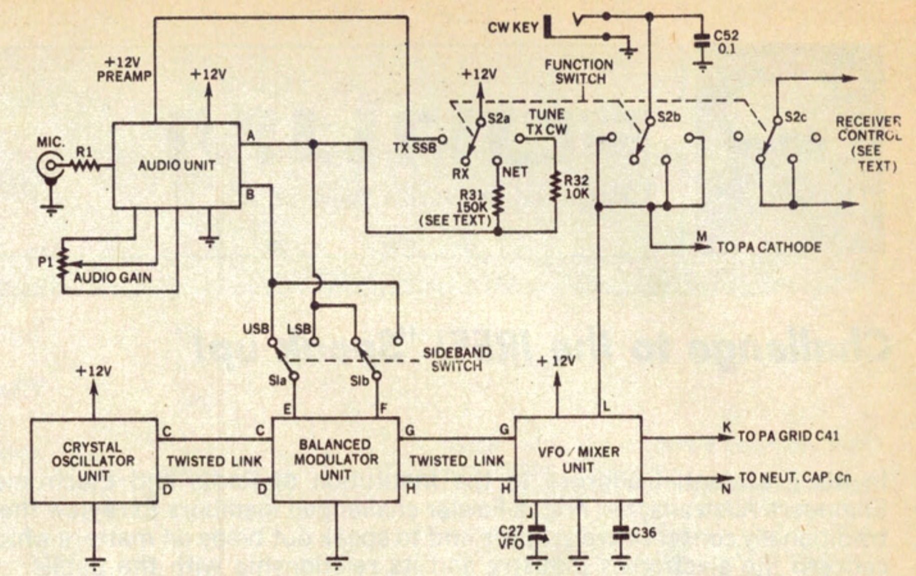


Figure 13: A diagram showing all the interconnections between modules.

across the socket can then be mounted.

The transformer can now be positioned. All the remaining components can then be mounted and the point-to-point wiring completed.

The printed boards of the prototype are mounted by using a solid copper wire mounting clip as shown in figure 14. The boards are held close against the tin by this method. Make certain that these mounting clips are soldered to the EARTH parts of the printed board and not to some other part!

Holes have been provided on the board patterns for conventional screw mounting if this is preferred.

The slug in coil L7 is removed and a 4-inch length of discarded plastic knitting needle is glued to it (I used Araldite). A hand-drill is used to bore a hole in the chassis under L7. This is most easily located by drilling through from the coil side down the axis of the coil. The knitting needle is then inserted through the hole. This enables the tuning of L7 to be accomplished from above the chassis. This control is labelled "mixer peak" on the top of the chassis. This simple method avoids the problem of providing and setting up wide-band tuned circuits. Make sure that the position of the mixer and VFO board is such that this knitting-needle control does not foul any above-chassis components (e.g. L10).

It is hoped that the photographs will assist with providing other details necessary for construction.

The third and final article in this series will describe the testing and adjustment of the rig. It will describe an audio oscillator, a dummy load and a wavemeter, all very useful aids around the amateur shack, and will also give coil winding details and a voltage table. Details will also be given for a modification to permit AM operation.

(To be continued)