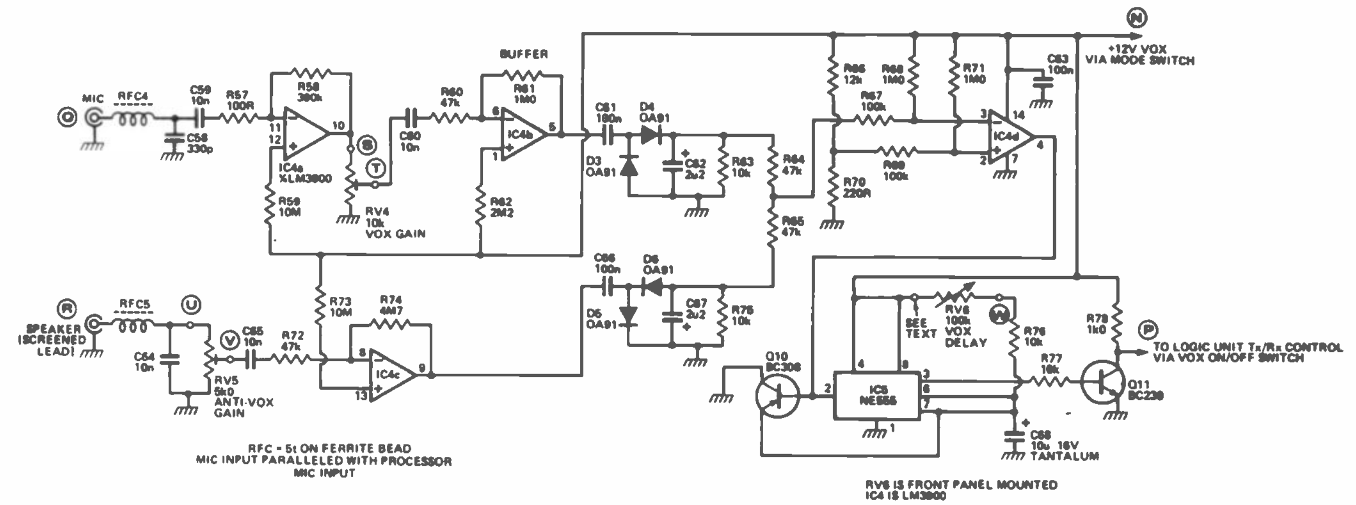

Omega: VOX unit circuit

Note that some of the mylar capacitors use two values in parallel to get the required capacity. This applies to the 27n (22n + 4n7) and 12n (10n + 2n2) capacitors – the two are marked as 'a' and 'b' on the layout diagram. Watch polarity of electrolytics. For Tantalum capacitors (usually in blue or red encapsulations), if the polarity is not marked, then either the longer lead is +ve, or the right hand lead with the identification facing you is +ve. Reversing these capacitors can have all sorts of strange effects (other than possible explosion!).

4. Finally, insert and solder all semiconductors, noting orientation of ICs and cases to agree with the layout.

Finally check out everything again to avoid later problems.

Testing

The unit can be partially tested at this stage if you have an AF Oscillator and scope, otherwise it's best to wait until the whole unit is finished. Testing at this stage is accomplished by:

1. Connect the two points K together, and power up with +12V. Current consumption should be around 45mA. Apply +12V to point G – point L should go to +11.5V.
2. Connect an AF generator at 1kHz to the mic input and a scope to point J (AC coupling on the scope). Set RV2 and 3 to mid travel. Inject a few millivolts of AF signal, and check that the scope output amplitude limits very quickly as the input voltage is increased. Leave the input voltage at about 25mV. Adjust RV3 so that the circuit just starts to limit – as shown by the peaks of the sine wave just starting to flatten. This sets the processor to minimum distortion conditions. You can also check that the output falls off rapidly above 2.5kHz, and below 300Hz.

If you have none of the above equipment, just leave the two presets set to mid travel for the moment, after checking Step 1.

RF/VOX PCB

This is a double sided PCB, with the top foil acting as the earth plane. As usual with all our RF circuits, keep the leads as short as possible.

1. Insert and solder the 22PCB connection pins, noting that two of them are acting as earth connections to the underside (adjacent filter, and VC1/2).
2. Insert and solder IC4 & 5, watching orientation.
3. Starting at the top left hand corner of the PCB, work round the board with the components. This should make it easier to insert components where space is tight, providing you look ahead all the time, noting where earth connections have to be made. Watch the following points: C67 has its *positive* lead connected to earth; the four diodes are slightly fragile, so take care when bending their leads; RFC1 and 2 are green and coded 101 + a letter; T1 needs 12cm of wire for the 6 turn primary, and twice 8cm for the secondary (see July HRT if you are not sure of how to wind these cores) – try to wind this transformer as symmetrically as possible to aid carrier balance; RFC4/5 need 10cm wire each.

Testing

You are advised to check this board out, and its use with the AF board as far as possible before mounting it all into its box, as this could save a lot of problems later. For full testing, a scope suitable for 10.7MHz is needed – otherwise the RF side will have to be checked 'on-air' later. Proceed as follows:

1. Connect +12V to point C. Current consumption should be around 5mA. Add +12V to point A,a and the current should rise to about 9mA, showing the USB oscillator is working. Likewise, power to point C should also give 9mA. If you have a counter available, connect it to point D and set the USB oscillator to 10.7015MHz, and LSB to 10.6985MHz. If you have no counter, you should set these on frequency at this stage with the unit connected to the CIFPU. Disconnect the link between points N & P on the latter, and connect D on this unit to M on the CIFPU via coax. Then adjust the trimmers for correct reception of the appropriate sidebands. The trimmers will be about ⅔ meshed although this will vary.