Modern High-performance Narrowband Equipment for 10GHz

Part 1 The G3WDG-001 Multiplier / Amplifier

Designed by

C.W. Suckling, G3WDG and G.B.Beech

Modern high-performance narrowband equipment for 10GHz

By C.W.Suckling, G3WDG, and G.B.Beech.

Part 1 - General design features and the G3WDG-001 multiplier/amplifier

Introduction

Over the past few years there has been a trend away from traditional waveguide techniques for amateur narrowband equipment at 10GHz. Equipment using GaAs FETs in microstrip circuits offers higher power levels, better receiver performance and is arguably easier to build. Construction techniques follow lower frequency practice rather than "plumbing", although care and attention to detail are still required if good results are to be obtained.

A number of GaAs FET-based designs for 10GHz have appeared in amateur literature, for example [1]. These have tended to use relatively expensive GaAs FETs and the cost of construction has been too high for many amateurs. Recently a large quantity of GaAs FETs have become available on the UK surplus market at a fraction of the cost of "new" devices. The GaAs FETs in question were manufactured by the Plessey 3-5 Group for use in 11GHz satellite TV LNBs and have excellent performance. The low cost of these led the authors to develop a number of new designs for 10GHz.

The designs are intended for home construction without the need for either difficult construction methods or elaborate test equipment. They have all been duplicated with relatively little difficulty by a number of independent constructors. Wherever possible, low cost components have been identified and designed-in, but in some cases it has been necessary to use more expensive components. The specified parts MUST be used throughout or the hard work put in by the designers to make the designs reproducible will have been wasted! All the special components, with the exception of the GaAs FETs, are available from the RSGB Microwave Committee Components Service. Problems were encountered during the design-proving phase when some constructors had not used the correct grade of GaAs FETs in some locations. The different surplus GaAs FETs available are NOT interchangeable!

Three designs will be described, all of which require drive input of 10mW or more in the 2.5 to 2.6GHz region for full output at 10GHz. The exact frequency depends on the particular application. A suitable oscillator/multiplier, the G4DDK-004 design [2] has already been described, giving the required level of output in this range, and PCBs for that design are available through the RSGB Microwave Committee Components Service (G4DDK PCB 004). Subsequent to the original article, a number of minor modifications were made to some of the circuit values [3] which raised the output to 10mW or more, sufficient to drive the present designs to the required output. If the output level of the DDK-004 board is measured at 10mW or greater, then it will be possible to omit the MSA0504 MMIC amplifier stage in the designs.

The three designs to be described are:

1. A x4 multiplier/amplifier chain (G3WDG-001) which can provide 50 to 100mW output anywhere in the 10 - 10.5GHz band. It can be used as the basis of a simple CW/FM narrowband transmitter, a beacon or personal signal source, as an ATV transmitter or as a packet radio link transmitter.

2. A down-converter (receiver), G3WDG-002, to 144 - 146MHz incorporating a x4 multiplier chain to generate the local oscillator signal, a dual-diode mixer and two stages of low-noise pre-amplification before the mixer. The design also incorporates a low-noise post-mixer amplifier at the intermediate frequency. The front-end noise figure of several prototypes have

been measured at less than 2.5dB. It is possible to improve this figure by using an external pre-amplifier.

3. A linear up-converter (transmitter), G3WDG-003, from 144 - 146MHz to any 2MHz segment in the 10GHz band. It incorporates a x4 multiplier chain, a GaAs FET mixer and four amplifier stages to reach an output power of at least 50mW. A further power amplifier stage is under development using a Mitsubishi MGF1601 GaAs FET and the prototype is giving over 200mW output.

By providing the 002 and 003 modules with a common 2.556GHz local oscillator source and suitable transmit/receive changeover arrangement, the result is a complete "state-of-the-art" linear transverter.

It is a good idea, for those not yet skilled in the art of microwave PCB construction, to work through these module designs in the order described since this represents a steady progression from a relatively simple to quite a complex design. If you can successfully build and align the G3WDG- 001 design, then you are well on the way to building your own state-of-the-art high performance receive converter or linear transverter at a fraction of the cost of comparable commercial equipment.

General circuit features and components.

All three designs are built on ptfe-glass board and, in the main, surface- mount "chip" devices (SMDs) are used, although some more familiar "ordinary" components are also used. The "heart" of the units is the widespread use of GaAs FETs as active multipliers, amplifiers and, where required, as mixers. Microstrip circuitry is used to provide the correct operating impedances for the GaAs FETs and the circuits have been designed to cover the whole of the 10GHz band from 10.0 to 10.5GHz. A reliable method for grounding the source leads of the GaAs FETs was developed to ensure that the designs would be reproducible. Earlier attempts using copper foil "wrap-arounds" failed because the inductance of such connections was too variable.

Where high selectivity is required, to discriminate between harmonics or to reject image frequencies, this is provided by the use of small "pill- box" tuned cavity resonators soldered to the board. Coupling from the microstrip lines into and out of the resonators is accomplished by the use of probes. This technique has been common in German amateur microwave designs for some time, for example [4], and avoids the use of critically dimensioned and spaced printed microstrip filters which are almost impossible to make with enough accuracy. It will be noted that a high drive oscillator frequency has been chosen (around 2.5GHz), also in order to minimise the stringency of filtering. The drive source chosen gives output at the required level, with all unwanted products at least -40dBc or better: this minimises the filtering requirement at the final signal frequency and makes the use of single, simple cavity resonators possible wherever such selectivity is needed.

Similar principles have been adopted for all three designs, each GaAs FET amplifier stage providing a gain of about 10dB. Matched input and output circuits are realised by the use of microstrip lines. Rather than attempt to etch very narrow (high impedance) microstrip lines, where these are necessary, easier construction results from the use of short lengths of thin wire soldered to the lines and pads on the surface of the board.

It is recommended that the PCBs available from the RSGB Microwave Committee's Components Service be used for these designs. Both the board material and the dimensions of the microstrip lines are critical to the success of this type of circuit. The other critical components, such as the ceramic chip capacitors, resistors and the resonators, are also available. Virtually all the other components are available from easily accessible amateur sources.

ONLY the recommended components should be used and only first-grade KNOWN components employed -substitution from the "junk-box" or components "salvaged" from other microwave equipment is just not acceptable!

It is strongly recommended that, whichever module is to be constructed, the PCB is installed in a tin-plate box or an alternative, specially made sheet-metal (brass or copper) enclosure of similar size and form. By so- doing, not only is the somewhat flexible board housed rigidly, but is also well screened and thermally insulated to some degree. The finished, boxed unit(s) should be housed in a rigid outer case to provide mechanical and thermal stability - the "boxes within boxes" approach which has been advocated for high performance microwave equipment, almost regardless of frequency.

The use of other than SMA connectors for input and output is not recommended. The 12V power supply (or any other ingoing supplies) should be well decoupled by 1nF to 10nF solder-in feedthrough capacitors or Filtercons. The power supplies must be stabilised to the voltages given in the circuit diagrams: if these voltages are exceeded, or the gate bias voltage fails, the GaAs FETs can be damaged, if not instantaneously destroyed. By incorporating resistors in the drain circuits a degree of current-limiting protection is afforded. Nevertheless, it is well worth spending time on this aspect of the circuits, using only generously rated and reliable components in the bias circuits.

Care and attention to detail is essential AND your soldering techniques must be good! Components should be mounted in the order given and the GaAs FET devices should always be the last components to be soldered into place, taking the usual precaution of grounding together the constructor, the body of the soldering iron and the case/groundplane of the PCB whilst soldering them in place. In this way the risk of damage by static discharge is minimised or eliminated.

The G3WDG-001 multiplier/amplifier

The circuit is shown in Fig. 1, the layout of the board and components in Fig. 2 and their values in Table 1.

As with all the units to be described, drive at 2556MHz (for output at 10224MHz) or 2592MHz (for output at 10368MHz) is required at a level of 10mW or greater. This is supplied by a suitably crystalled G4DDK-004 oscillator multiplier strip, the crystal being a fifth overtone HC18/U type in the range 106 to 108MHz. To accommodate the varying output levels of the drive source (variations being caused by individual component tolerances and the accuracy of construction and alignment), provision is made for the inclusion of a broadband MMIC amplifier, using an Avantek MSA0504 device. This is capable of providing more than sufficient gain to drive the GaAs FET multiplier, F1 (Fig.1), adequately at the chosen input frequency. If the output of the G4DDK-004 driver is measured at 10mW or more, this MMIC amplifier stage can be omitted and the gap in the microstrip line occupied by the MMIC "patched" with a small piece of copper foil soldered across the gap. L1, R1 and C3 may be omitted, but C1 and C2 must be fitted. If the MMIC is used, it will be necessary to make a hole large enough to take the body of the MMIC so that it can be mounted on the board with minimum lead length -in a manner similar to the BFR91/6 transistor package used in the driver board G4DDK-004.

The output from the MMIC is matched to the input impedance of the multiplier FET by a "lumped element" network consisting of L2/L3. L3 is also used to feed the negative gate bias to the FET from RV1 which sets the optimum operating bias for the multiplier. The output circuitry at the drain of the FET consists of a series resonant circuit at 2.5GHz followed by some microstrip matching elements at 10GHz. A series resonant circuit is formed by L4 and a printed capacitor (identified by the cut corner). The function of this circuit is to "short circuit" the input frequency to ground, which improves multiplier efficiency considerably. The drain bias circuit consists of a quarter-wave choke, L5, connected to a quarter wave stub. The tip of the stub is at very low impedance at its resonant frequency (10GHz). This is transformed by L5 to a very high impedance. In this way there is virtually no disturbance to the signals on the microstrip line from the bias network. This type of bias network is fine at its operating frequency but needs additional decoupling at lower frequencies. This is accomplished primarily by R2/C5 which load the drain of the FET resistively at low frequencies, giving broadband stability. C10 is used for further decoupling at very low frequencies. This bias configuration is used throughout the 10GHz designs, except that in many locations the quarter-wave stub is replaced by a triangular element. This has the same properties as a stub ie. the tip of the element is at very low impedance but is very much smaller, resulting in a more compact layout. R7 is used to set the drain voltage to an optimum value for best multiplier performance.

The output from the multiplier consisting of several harmonics, but principally the wanted fourth harmonic of the drive frequency, is fed into FL1, the resonator filter, by means of a probe. Filtered output is coupled to the gate of the first amplifier, F2, by means of a second probe. The resonator is tuned by means of an M4 screw, locked in place with a lock-nut after tuning to resonance. The resonator body and tuning screw can, with advantage, be silver plated. Not only does this increase the Q of the cavity significantly, but also makes the cavity much easier to solder to the board.

GaAs FETs F2 and F3 form a cascaded two stage amplifier yielding about 20dB gain, each with gate bias and drain bias arrangements similar to those of the multiplier stage. DC blocking and RF coupling in the drain circuits is accomplished by means of 2.2pF chip capacitors which MUST be designed specifically for such frequencies eg. ATC series 100 or 130. The ordinary chip-C's used for lower frequency coupling or decoupling are definitely not suitable in these two positions.

The drain supply of the three GaAs FET stages must not exceed 8V measured at the supply rail, otherwise their ratings may be exceeded and they will become expensive, fast-acting fuses! Similarly a gate bias rail at -2.5V is needed to set their operating points. Two simple IC regulator circuits can be built onto a small piece of fibreglass board housed in the groundplane compartment of the box. A suggested circuit is shown in Fig. 3 and a layout for the inbuilt regulators is shown in Fig. 4. Note that the PCB layout shown for the regulator board requires the components to be mounted on the track side, as if the components were surface mount devices. Thanks go to G4FRE for this design.

Construction of the G3WDG-001 Multiplier/Amplifier

It is strongly recommended that the following procedures are followed in detail and in the order described for both this and the subsequent designs:-

1. If the MMIC amplifier is to be used, carefully drill a hole, 3.7mm diameter in the middle of the gap in the microstrip lines identified from Fig. 2 as the position of the MMIC. It is suggested that a small pilot hole is drilled from the track side which is then carefully opened to the required size by drilling from the groundplane side. Care will be needed to avoid tearing the copper of the groundplane. If this amplifier stage is not to be used, then simply bridge the gap with a piece of copper foil, the same width as the lines, soldered across the gap.

2. Ensure that the copper of the tracks and the groundplane is clean and bright. Various means of achieving this are described later in connection with local cleaning just before critical soldering operations. Where light tinning is required, use a small soldering iron and

very fine solder. If too much solder is applied accidentally, remove excess using a solder sucker - before trying to solder the small chip components in place!

3. Fit grounding PCB pins and filter locating pins (see later) and solder in place. Lightly tin around the edge of the groundplane.

4. Solder the filter into position. Leave the tuning screw and lock nut into position to avoid unwanted debris accidentally falling into the cavity.

5. Locate the PCB into its box and trim to a neat fit if needed, particularly in the corners of the box where there are joints. The PCB material will cut quite easily with a sharp scalpel blade and straight-edge. Locate the groundplane 17mm from the top of the open box and mark its position. Locate and mark the SMA socket centre pin clearance holes. Drill the holes and de-burr. Locate, drill and de- burr holes for any feedthrough components needed for power supplies. Tack-solder the corner seams of the box and make sure that the lids are a neat fit. Adjust as necessary. Check also that the board will fit neatly. When satisfied, solder the corner seams fully. Solder the SMA connectors and the feedthroughs in position.

6. Relocate the PCB so that the input and output tracks touch their respective socket spills, tack-solder the PCB in place and, when satisfied that it is correctly located, solder all round the groundplane and solder the SMA socket spills to their respective tracks. Note that the +8V rail is very close to the edge of the board and care must be taken to avoid an accidental short circuit to the wall of the box whilst soldering the board in place. This completes the mechanical construction of the module.

7. Fit inductors L1 - L9, as specified in the parts list (Table 1), into position, ensuring inductors L2 - L9 lie flat to the board.

8. Fit all chip components using the mounting techniques described later. You will need a pair of fine-pointed tweezers to handle these small devices and, maybe, the assistance of a magnifier!

9. Fit all components which have leads, ensuring that static-sensitive devices (ICs, FETs) are put on the board last of all to minimise the risk of damage to the devices.

Note:- it is best to apply the supply voltage to the board BEFORE fitting IC1 and the FETs, to check that both the +8V and -2.5V voltages are present and correct on the respective tracks/pins. On completion of this test, disconnect power and solder in the devices only if everything checks out correctly.

Individual "build" techniques.

1. Pcb-pin grounds.

Clean the groundplane on the rear face of the PCB using an RS Components PCB eraser (Part number 555-303), or similar mildly abrasive pad - a really clean surface enhances solder flow. Tin around the hole on the groundplane side. RS Components PCB pins, part number 433-854, 1mm diameter with a head diameter of 1.5mm, are used wherever there is a need for through-board grounding. Other makes of pin, eg. Veropins, could be used provided that their size is the same.

Place the PCB-pin in the hole with the pin head on the track side and the body of the pin sticking up through the groundplane, with the exception of the three filter-locating pins (see below). Place the head of the pin on something hard and flat and press the board until the head butts up against the track side of the board.

Solder by starting with the iron at the top of the pin; tin the pin generously and, while applying more solder to the joint, flow the solder down the pin and onto the groundplane to ensure good pin to groundplane contact. Trim the pin back using flush-cut cutters.

2. Fitting the filter

This is potentially the most difficult soldering operation on the board! Note that cleanliness is a MUST for this operation and the rear face of the board ie. the groundplane, should be "shining clean". The surface should be cleaned carefully using the RS Components PCB eraser or other similar mildly abrasive pad and, if possible, degreased with an aerosol cleaner eg. RS Components Part Number 567-660 or 554-838. Details of the cavity are given in Fig. 5(a), together with the dimensions of the coupling probe pins. First prepare and clean the PCB, then fit the three PCB-pins which mark the filter cavity position. These pins are fitted from the groundplane side through to the track side. Solder the pins to the pads provided on the track side and cut off excess pin length after soldering. The board is now ready to take the cavity filter.

Pre-heat the cavity with its tuning screw and locknut assembly in position. Heat it by placing on a hot plate (eg. a 3 to 6mm thick sheet of aluminium placed over a gas ring) and heat until 60/40 tin/lead solder melts easily on touching it to the cavity wall near the base (open end). Quickly transfer the hot cavity, using pliers to grasp the tuning screw, to the board, position it between the three guide-pin heads on the groundplane side of the board and apply fine (22 SWG or finer) solder at the junction of the board and filter to fix the cavity in place. Ensure the cavity does not jump outside the guide pins whilst soldering, ensure a continuous small fillet of solder all round the cavity, but do not apply too much solder. Allow the cavity to cool without disturbing it. When it has cooled fully, fit the two PCB-pins which probe through the board and into the cavity, having pre-cut them to the length shown in Fig. 5(a).

3. Inductors

For inductors using enamel covered wire (ECW), cut the required length of wire then scrape/chip the enamel from the last 1-2mm of each end using a scalpel blade. Tin each end.

For inductors using one strand of a standard multi-strand wire, tin one end and fit to the board as shown in Fig. 5(d): solder first at position 1, then at position 2 as close as possible to the apex of the triangle, then at position 3. If any excess wire remains at 1 or 3, trim off carefully with a scalpel blade.

4. Chip components

To fit chip components across two circuit tracks or pads, adopt the following procedure for best results: (see Fig. 6)

a. Lightly tin one of the tracks or pads.

b. Fit component and reflow solder to make a solder fillet at the tinned side - the tip of the tweezers may be used to hold the chip in position whilst the solder solidifies. Use as little solder as possible to form a very small fillet.

c. The component should now be secure: tin the other track and make a solder fillet on the second side of the chip component to complete the mounting.

d. Resolder the first joint if required, using a little fresh solder.

This procedure ensures that the components are flat to the board, good contact is made and the best circuit performance is achieved.

5. Static-sensitive components

Components such as ICs and FETs should always be fitted last to minimise the risk of static damage. The GaAs FETs have the gate lead bevelled for identification, as shown in Fig. 5(b). Grounding of the two FET source leads is via PCB-pin ground "pads" fitted as shown in Fig. 7. Cut the source leads to minimum length, but note that before handling static- sensitive devices, you should make sure that you and the handling implements (eg. tweezers) are grounded together: it is often a good idea to work on a grounded sheet of aluminium foil spread on the work surface, resting the wrists on the foil, with the board and implements also on the foil when not in use. You may find, if using surplus GaAs FETs, that one source lead is already trimmed short. Cut the other to a similar length.

Lightly tin the source grounding-pins and the ends of the lines - the same remarks about cleanliness and minimum amounts of solder apply here also! Place the trimmed FET as shown in Fig. 7 and reflow the solder on one of the source leads, then the other. Push the gate and drain leads down flat onto the board as close to the FET as possible, cut off excess lead length carefully with a sharp scalpel blade, then solder them down to the respective tracks, making sure that the device is orientated the right way round!

Alignment with simple test-gear

Once completed, the PCB should be carefully examined for poor joints, accidental solder bridges and other forms of short circuit. Once satisfied that all is well, the alignment procedure may begin. You should already have checked before mounting the IC (if used) and FETs that the correct supply voltages will appear on the positive and negative supply rails when a 12V supply is connected to the input feedthrough capacitor.

1. Turn the bias potentiometers RV1, RV2 and RV3 fully clockwise so that full (cutoff) bias will be applied to the gates of the three GaAs FETs when power is applied.

2. Insert a multimeter in series with the +8V supply between the regulator output and the +8V rail and set it to, say, 500mA full scale deflection.

3. Attach a suitable load/power indicator to the output socket. This might consist of an SMA to Waveguide 16 transition, variable attenuator, wavemeter and detector connected as shown in Fig. 8. Here's where some of your older WG components come into their own!

4. With no oscillator drive applied to the input socket, apply +12V to the power input feedthrough. Current consumption should be no more than a few microAmps of "leakage" current. Switch the range of the multimeter as needed. If considerably more current is measured, look for short circuits or mis-connected or damaged components.

5. If all is well, adjust RV1 so that the current consumption rises to about 2 or 3mA.

6. Adjust RV2 for a further rise of 20mA in total current.

7. Repeat for RV3, again looking for a rise in current of a further 20mA. At this stage the current consumption should be of the order of 42 to 45mA.

8. Set the variable attenuator in the load/power indicator to minimum and detune the wavemeter well away from the expected frequency.

9. Apply drive to the input socket at a level of 10mW or more. Some output may be seen on the power detector meter and the current consumption may rise slightly.

10. Starting with the tuning screw just entering the cavity, slowly adjust the filter tuning screw inwards until the output begins to rise. Continue adjusting for rising power output whilst increasing the attenuation to prevent excess current in the detector diode. When adjusted for maximum output, check that the correct harmonic has been selected by using the wavemeter. The tuning range of the cavity is such that at least two, and probably three, harmonics of the 2.5GHz drive frequency can be peaked by the tuning screw - hence the need to check that the right harmonic has been chosen. Continue adjusting the tuning screw until you are certain that the correct harmonic has been selected and peaked. Lock the screw in position with the locknut.

11. Adjust the gate bias settings slightly to optimise output, starting with the multiplier FET, F1, and finishing with the output FET, F3. Readjust and lock the cavity tuning screw as necessary.

12. There is some spread of characteristics of the completed units, so the bias current settings given in 5, 6 and 7 above should be regarded as starting figures only. However, no stage should be allowed to pass more than about 50mA and the output stage should always work into a well matched load. Some current limiting protection is afforded by the 47R resistors in the drain feeds.

13. This completes the alignment: the power output should lie between a minimum of 50mW and a maximum of 100mW. It is not advisable to try to squeeze more power output than this, especially if the load is not too well matched although, if you can guarantee a good match, then more output can be obtained by changing R6 for a 10R resistor. However, this is rather "caning" the final amplifier and excess dissipation in any stage may lead to rapid destruction of the device concerned. You have been warned!

14. Some constructors have been able to obtain more power by tuning the output circuit of the final amplifier. This was done by sliding a small piece of copper foil (about 3mm x 3mm) up and down the microstrip line and soldering in position when the best point had been found. Don't forget to power-down before soldering the copper foil in place! In one case this increased the power output from 75 to 125mW.

Modulation and stability

The frequency stability of the drive source oscillator is of paramount importance since the crystal frequency is being multiplied by a total factor of 96. The drive source design G4DDK-004 was evolved with this purpose in mind and, with a good quality crystal (10ppm or better), housed as suggested, it will be found to give good stability at the 10GHz output frequency. However, stability can be improved still further by fitting a crystal heater, such as the Murata "Posistor" type PTH507B01BM500N016 which simply clips onto the crystal casing, is supplied with 13.5V nominal and, after a short warm-up, maintains a steady crystal temperature regardless of ambient conditions. The initial in-rush (switch-on) current for a "cold" heater is of the order of 0.5A but requires typically 25mA maintenance current once at operating temperature.

With regard to modulation, the simplest form of modulation is FM or FSK achieved most simply using the varicap modulator described in [5].

If either crystal heater or modulator are to be fitted to the drive source, make sure that their supplies (DC or audio) are well decoupled, where they enter the box, by means of additional solder-in feedthrough capacitors or Filtercons. Additionally, make sure that the modulator components are fitted directly across the oscillator inductor with minimum lead length and maximum rigidity! Unwanted stray coupling, either RF or hum, must be avoided if maximum stability and minimum sideband noise on the 10GHz carrier are to be achieved.

References

[1] "Redesigned 10GHz SSB Transverter", P. Vogl, DL1RQ. Dubus Technical Reports, 4/86.

[2] "An oscillator/multiplier chain for the frequency range 2.0 to 2.6GHz", S.T.Jewell, G4DDK. Radio Communication, August 1990, RSGB

[3] "An update on the G4DDK-004 board", S.T.Jewell, G4DDK. Microwave Newsletter, 08/89 (February 1990). RSGB

[4] "10GHz transverter module - portable", J. Dahms, DC0DA, Belgian VHF/UHF Convention, November, 1988.

[5] "'A High Quality UHF Source' RSGB Microwave Committee, Radio Communication, October 1981, RSGB

Acknowledgements

The authors wish to acknowledge the assistance and collaboration of several amateurs during the development and testing of these designs. In particular G4DDK, G4FRE, G4PBP, G3ZQU, G0BPU, and G3PFR.

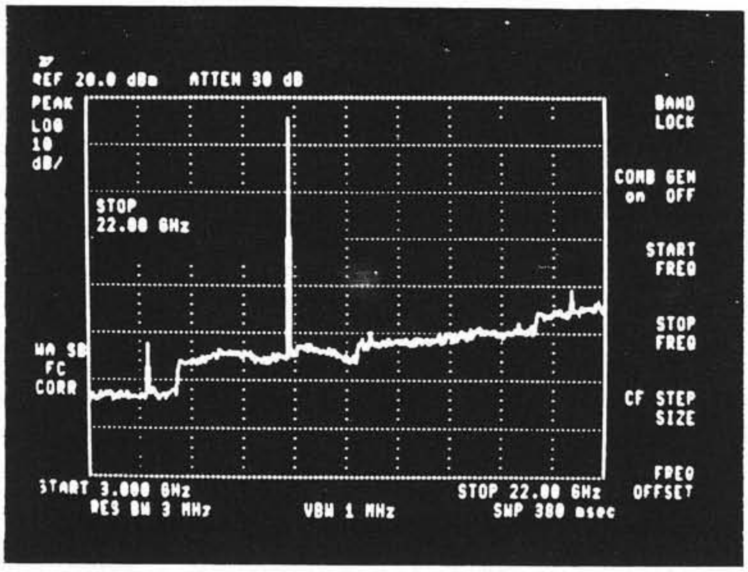
Typeset

G3PFR

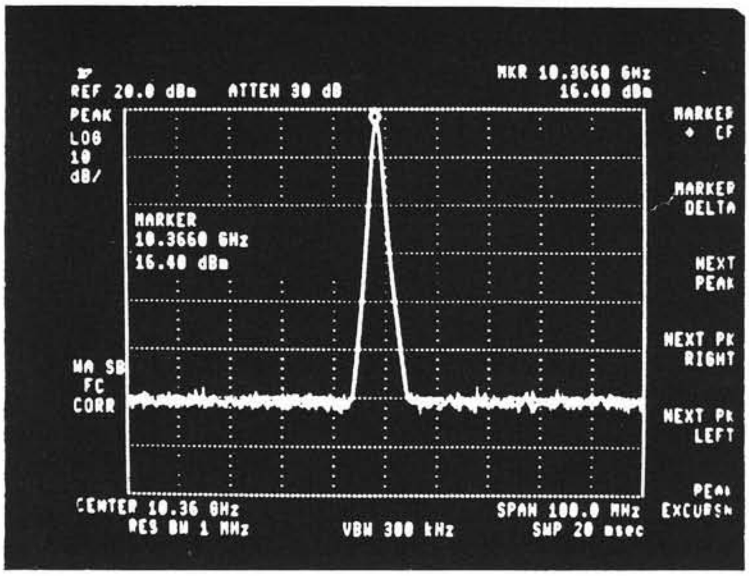
Graphics

G4DDK, G4FRE and G3PFR

Output spectrum of the G3WDG-001 multiplier board



Output between 3GHz and 22GHz showing the remarkably low level of sub-harmonics and harmonics. The rising noise level is due to the spectrum analyser, not the multiplier! The drive source was the the G4DDK-004 board at a level of +10dBm.



Close-in view of the output spectrum showing the clean output obtainable with this design, when driven by the G4DDK-004 oscillator, multiplier board

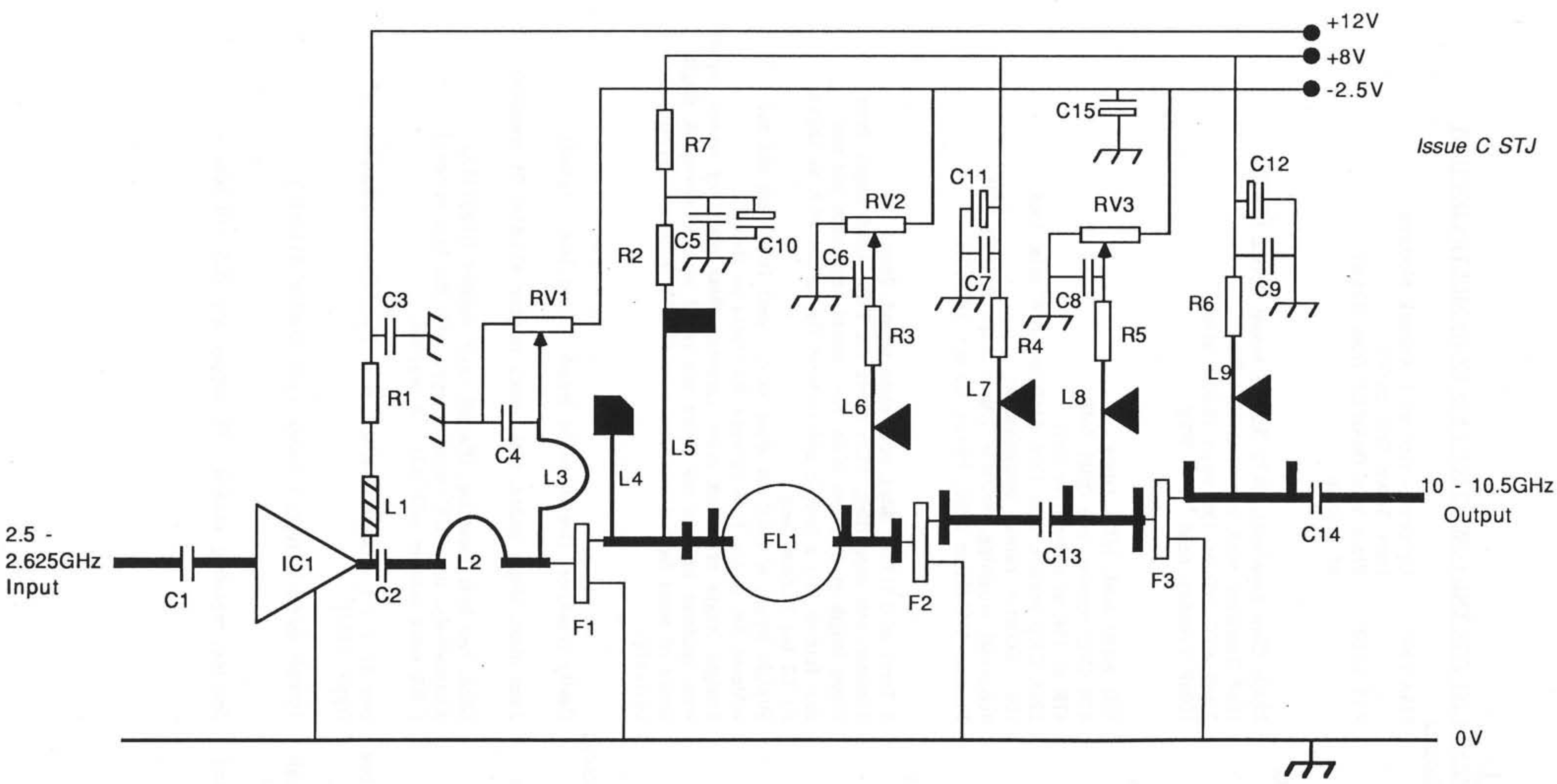


FIG 1 Circuit of the 2.5 to 10GHz multiplier unit G3WDG 001

Table 1

COMPONENT LIST FOR THE G3WDG 001 2.5 to 10GHz MULTIPLIER UNIT

Semiconductors

IC1	MSA 0504	Optional - (see text) Avantek Modamp from Wave Distribution
F1,2,3	P35-1108	Black spot GaAsFET from Birkett of Lincoln

Capacitors

C1 to 9	220pF Chip capacitors, 0805 Surface mount device (SMD)
C10,11,12	10uF Tantalum bead, minimum 10V Wkg
C13,14	2.2pF ATC 100 or 130 series (0.050" size)
C15	100uF Tantalum bead, 10 V Wkg

Resistors

R1	39R Axial lead, 1/2W rating
R2,3,4,5	47R Chip resistor, size 0805 SMD
R6	47R or 10R as above, (see text)
R7	220R Chip resistor, size 1206 SMD or 1/4W axial lead
RV1,2,3	10k skeleton cermet potentiometer horizontal mounting. Suitable types, Allen Bradley 90H, Bournes VA05H or 3309, Philips Comps. OCP10H.

Inductors

L1	6 Turns of 0.315mm diam. ecw, close wound, 2mm diameter, self supporting, 1mm above the PCB. Lead length 2mm.
L2	16mm length of 0.315mm diam. ecw, tinned 1mm each end and then formed into a hairpin and soldered flat to the PCB as shown.
L3	As L2 but 19.5mm long
L4,5	Straight length of 0.315mm diam. ecw, tinned 1mm each end and soldered flat to the PCB between the tracks as shown.
L6 to9	Straight length of 0.2mm diam. (approx) silver plated or tinned copper wire, soldered between the tracks and radial stub as shown. A single strand of braid from a miniature coaxial cable such as RG174/U is suitable.

Miscellaneous

FL1	Cavity resonator. Overall probe length 4.7mm (see diagram).
Veropins	1mm diam. single ended, (RS stock number 433-854; 25 required)
Sockets	SMA, two hole mounting. (Farnell stock number GE65137A). Alternatively, use PCB mounting type with the lugs removed. (RS stock number 403-752) : 2 required.
Tinplate box	size 37 x 111 x 30mm available from Piper Commmunications (type 7754)
Feedthrough capacitor	1000pF solder-in type, (Bonex stock number 032102)
Supply and bias	See text regarding suitable +8V supply and -2.5 volt bias.

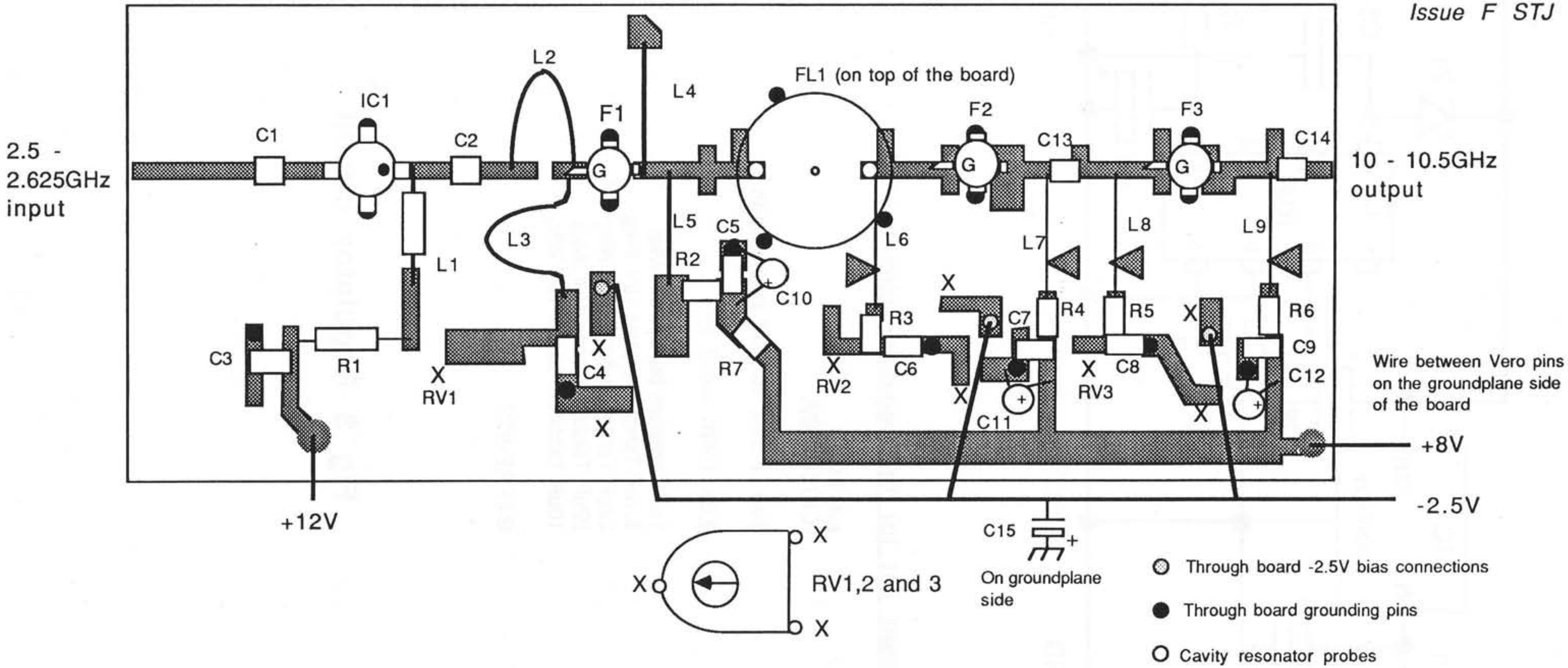
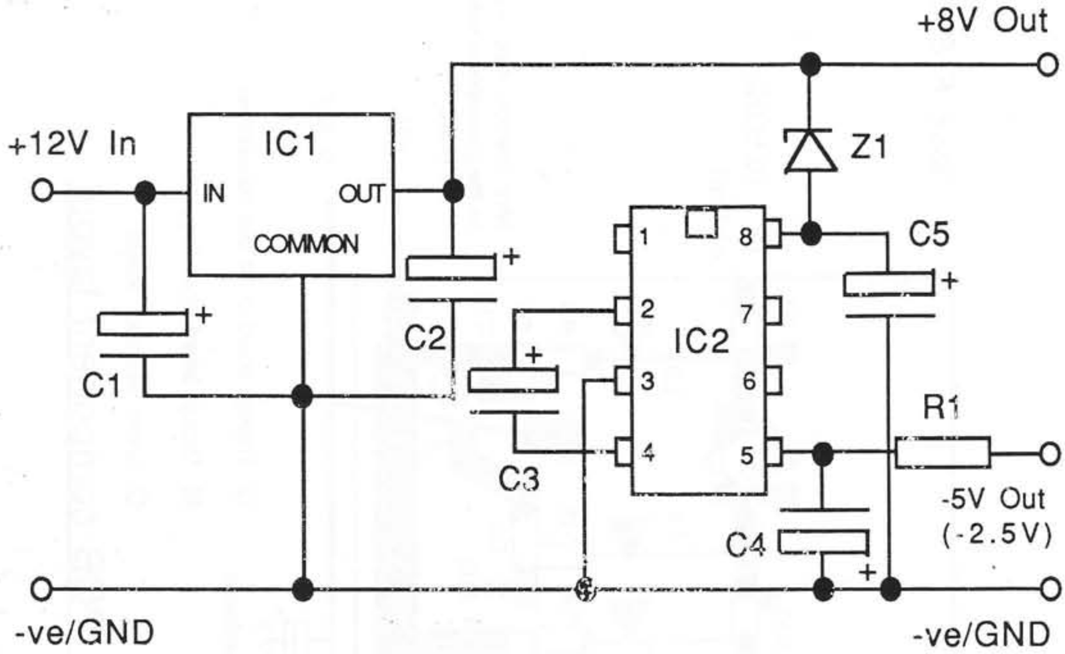


FIG 2 2.5 to 10GHz multiplier unit G3WDG 001 PCB component layout



Component list for the regulator circuit

IC1	uA7808
IC2	ICL7660PCA
Z1	3V0 or 3V3, 400mW zener diode
R1	3.3k 1/4W metal film
C1	1uF Tantalum bead, 16V Wkg
C2	0.1uF Tantalum bead,10V Wkg
C3	22uF Tantalum bead,10V Wkg
C4	22uF Tantalum bead,10V Wkg
C5	10uF Tantalum bead, 10V Wkg
PCB	G4FRE-023

Fig 3 Regulator circuit

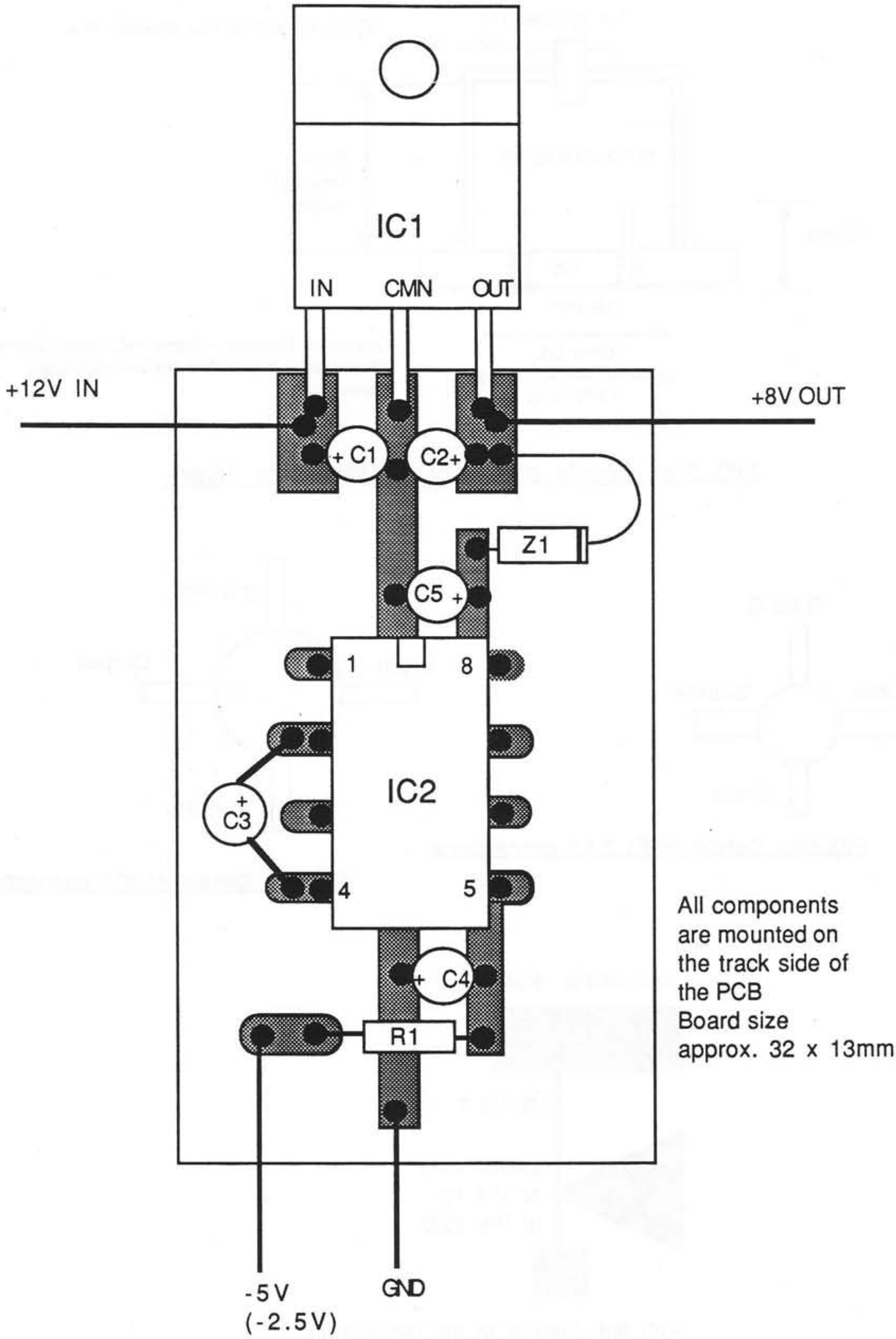


Fig 4. Regulator layout

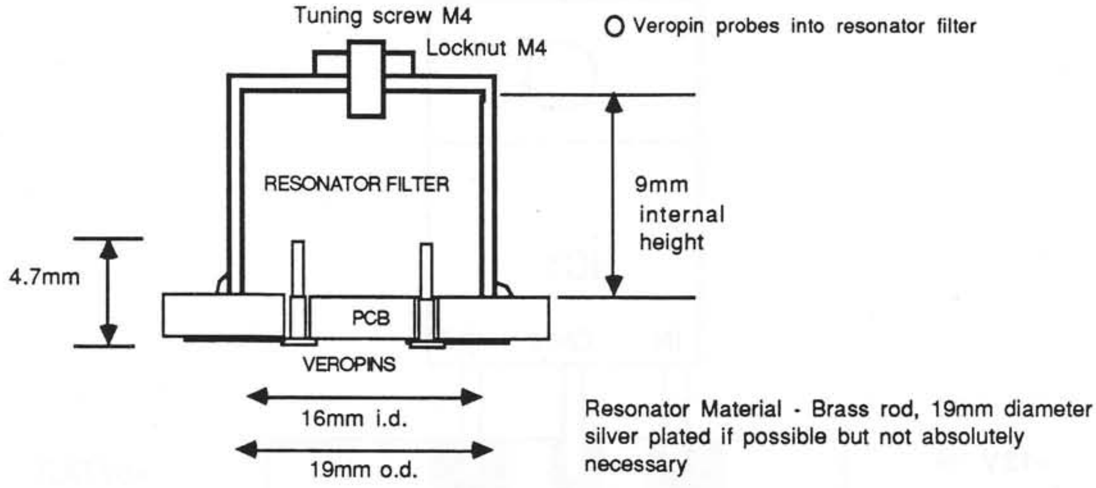


FIG 5(a) Details of the cavity resonator filters

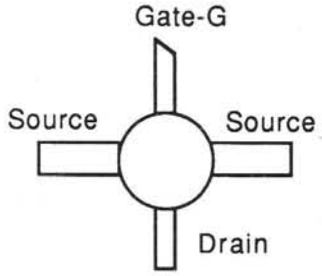


FIG 5(b) Details of F1,2,&3 connections

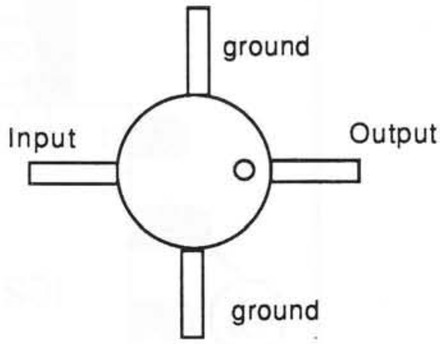


FIG 5(c) Details of IC1 connections

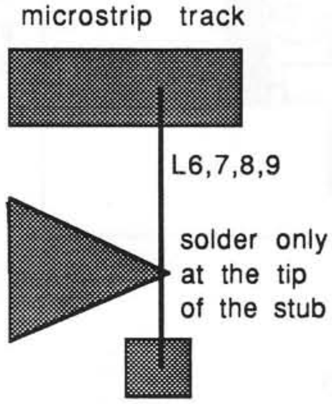


FIG 5(d) Details of the radial stub
connections to the bias chokes

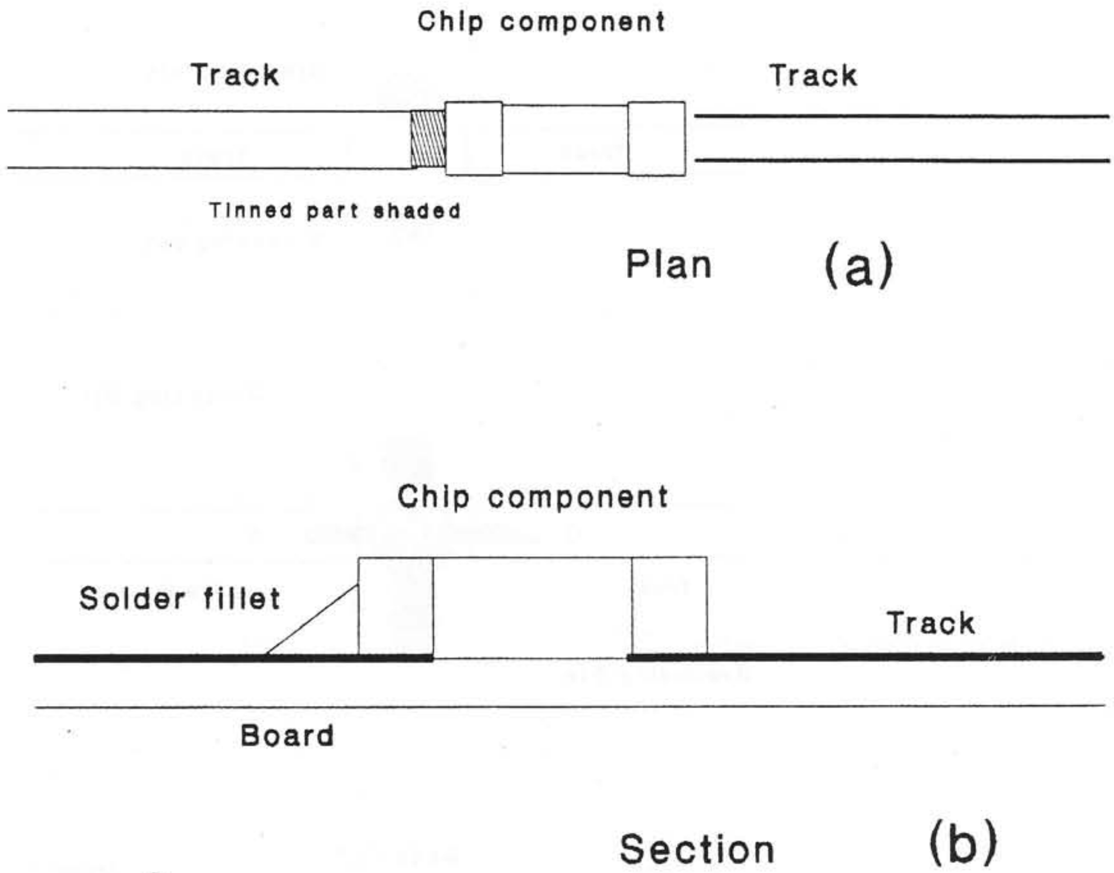


Fig. 6 Fitting chip components

G3PFR 07/90

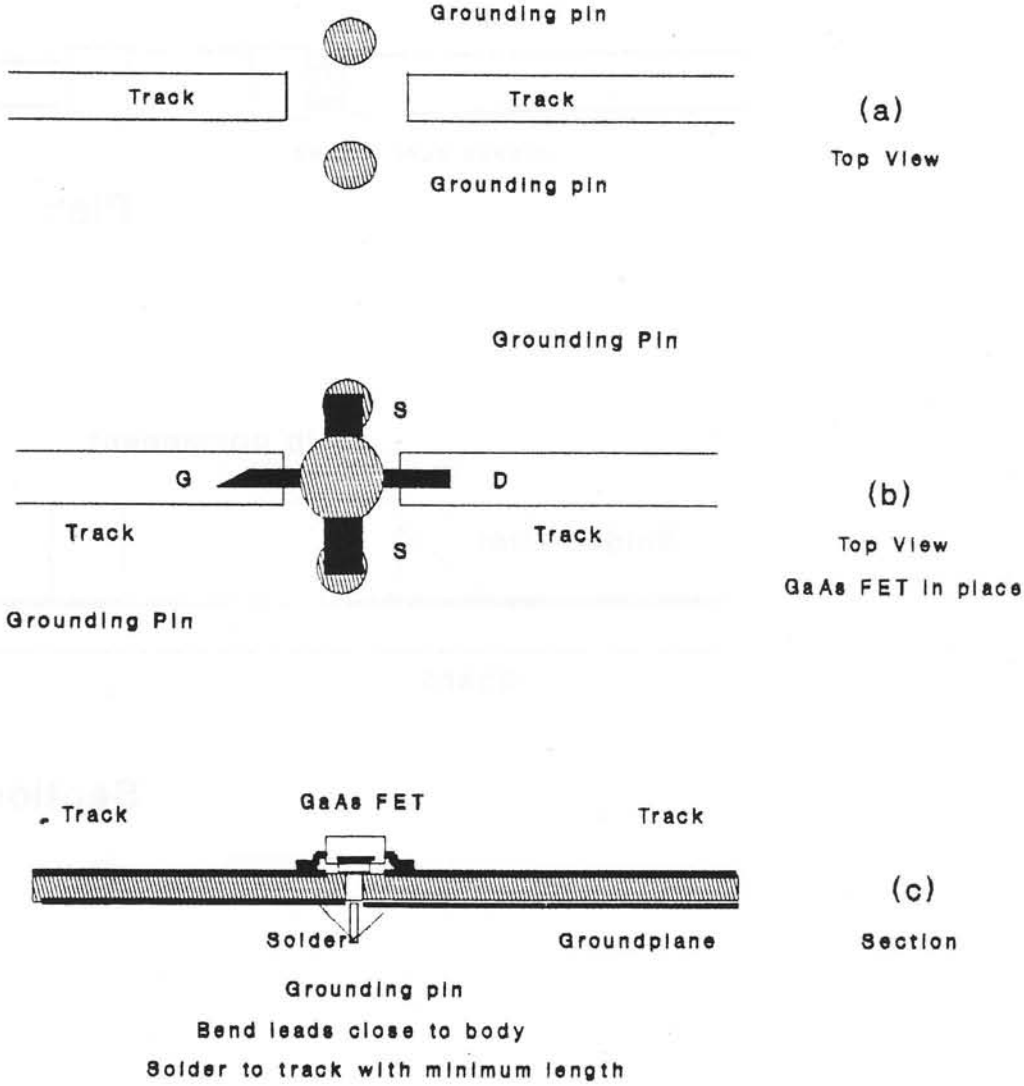


Fig. 7 Mounting GaAs FETs

G3PFR 07/90

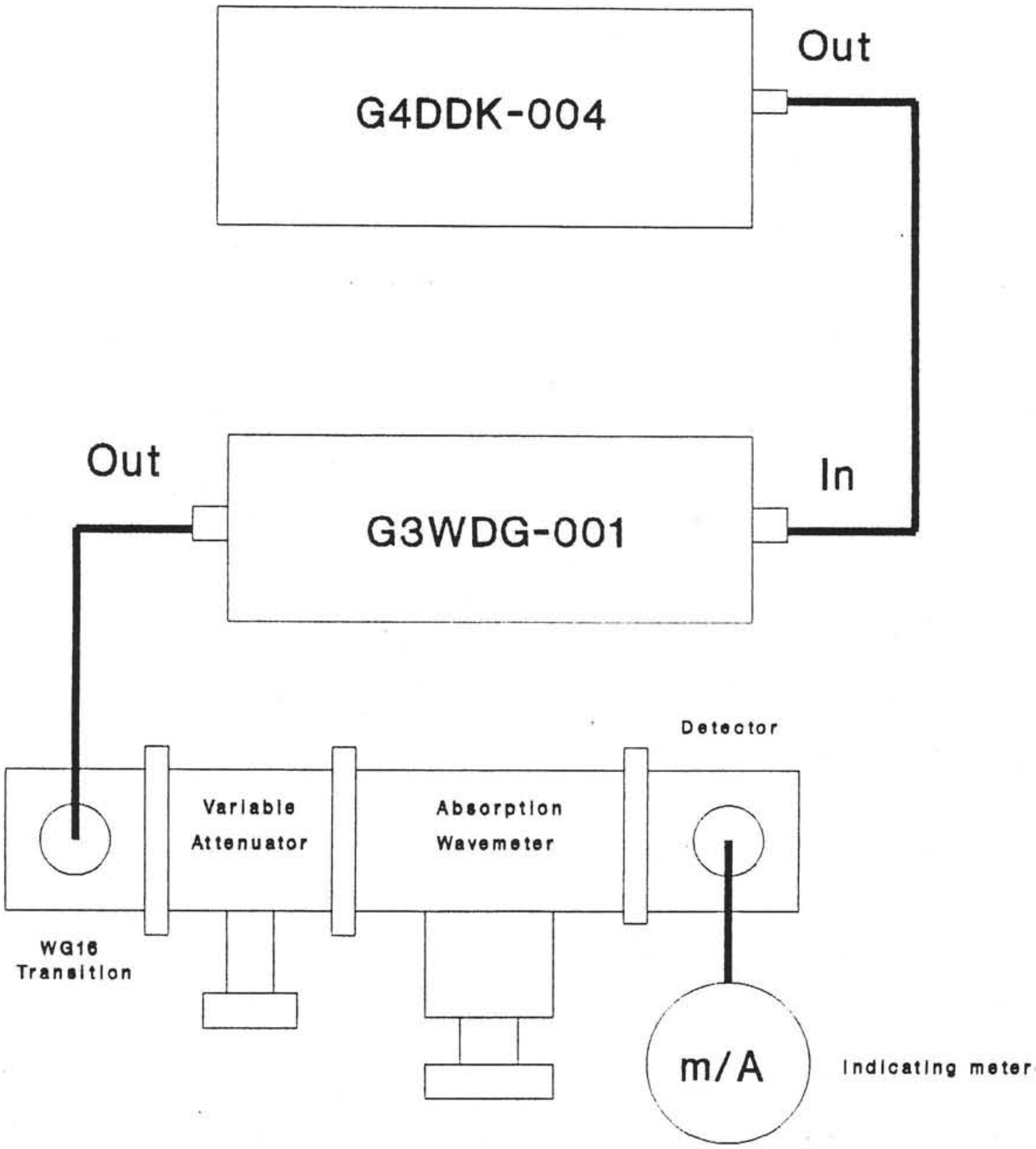


Fig. 8 Test Set-up

G3PFR 07/90