

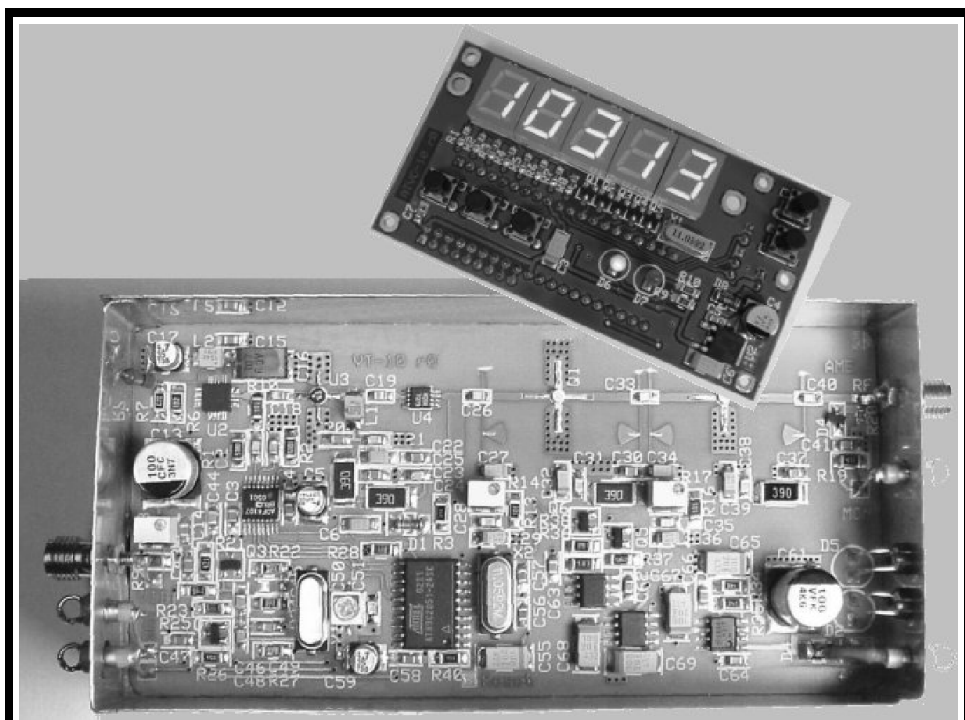


*A Publication for the
Radio Amateur Worldwide*

*Especially Covering VHF,
UHF and Microwaves*

VHF COMMUNICATIONS

Volume No.37 . Winter . 2005-Q4 . £5.20



**ATV transmitter with PLL for 10GHz
and Frequency input module**

Alexander Meier, DG6RBP

Practical Wireless, the magazine that brings you Amateur Radio & So Much More...



- *Reviews*
- *News*
- *Antennas*
- *Features*
- *Projects*
- *Radio Basics*
- *Bargain Basement*
- *Club News*
- *Vintage*
- *VHF, HF, ATV, Data*

**and much,
much more!**



Available from your newsagent or direct by calling
0870 224 7830 for £3.00

Why not take out a subscription for just £33 per year (u.k.)?



Contents

Aristoteles Tsiamitros	Systematic development of low pass filters using lines	194 - 205
Alexander Meier DG6RBP	ATV transmitter with PLL for 10GHz	206 - 216
Alexander Meier DG6RBP	Frequency input module for 10GHz ATV transmitter module	217 - 221
Gunthard Kraus DG8GB	An interesting program: Circuit simulation using PSPICE	223 - 238
André Jamet F9HX	Three probes for RF measurements without DUT disturbance	239 - 242
Wolfgang Schneider DJ8ES and Frank Peter Richter DL5HAT	Display unit for low power radio frequency wattmeter using an AD8362 from issue 3/2005	243 - 250
Gunthard Kraus DG8GB	Internet Treasure Trove	253 - 255

Plenty of interesting articles to keep you occupied over the festive season. Don't forget to renew your subscription for 2006. Once again postal price increases have caused a small increase in next years subscription.

I have been working on two new RSGB books this year. They have been published ready for your Christmas stocking. The 8th edition of The RSGB Handbook contains my chapters on VHF, UHF and Microwave transmitters and receivers plus Microwave antennas, see page 222. Microwave Projects 2 is a collection of interesting articles that I have accumulated from around the world, some from VHF Communications but also many more, see the inside back cover.

Merry Christmas and a Happy New Year. 73s - Andy



K M Publications, 63 Ringwood Road Luton, Beds, LU2 7BG, UK

Telephone / Fax +44 (0)1582 581051, email : andy@vhfcomm.co.uk

web : <http://www.vhfcomm.co.uk>



Systematic development of low pass filters using lines

The Richards Transformation can be used to create low pass filters using lines in the familiar LC low pass filter format. You use short circuited, ungrounded stubs for the series inductors and open circuited stubs of a specific length for the by-pass capacitors. With the help of Kuroda-Identities, the short-circuited stubs, which can be difficult to create using striplines, are replaced by open circuit stubs, which are easy to produce. The procedure is explained in a practical way below. ANSOFT designer software SV (Student Version) is used to develop the filter.

1. The Richards Transformation

It is known from line theory that a short-circuited line with an impedance of Z_L and an electrical length of ϑ has an impedance of $Z_k = jZ_L \tan \vartheta$. A line without a load with an impedance of $Z_C = 1/Y_C$ has an admittance of $Y_o = jY_C \tan \vartheta$. For $\vartheta < 90^\circ$ a short-circuited line is inductive and an open circuit line is capacitive. The transformation introduced by Richards [1] transforms the frequency $\omega = 2 \cdot \pi \cdot f$ to the new variable:

$$S = j\Omega = j \tan \vartheta = j \tan \left(\frac{\omega \cdot p}{v} \right)_{ab}$$

Note 1: Here, p describes the physical length of the lines. The letter l is assigned to the standardised inductance taking account of the phase velocity.

Among the new variables, $S, Z_k = S \cdot Z_L$ and $Y_o = S \cdot Y_C$ have the same form as the impedance of an inductance, $X = s \cdot L$ and/or the admittance of a capacity, $Y = s \cdot C$ in the s plane ($s = j\omega$). The transmit function of the filter in the $j\Omega$ plane is equal to that of the LC filter in the $j\omega$ plane [2 - 4].

Due to the periodical nature of the tangent, the passband of the line filter periodically appears in the $j\omega$ plane. The period is:

$$F = \frac{v}{2 \cdot p} \quad (\text{Fig 1})$$

If, for example, you select an electrical length equal to 45° ($p = \lambda/8$), then the period amounts to:

$$F = \frac{\lambda \cdot f_0}{2 \cdot \lambda/8} = 4 \cdot f_0$$

The shorter you select the stubs to be, the bigger the period F becomes. The periodic passbands are pushed further away from each other. The low pass filter becomes more broadband (see also Figs 12 and 13). But the problem with this advantage is that the line impedances also change and can possibly become too



With f_0 the cut-off frequency of the LC filter and f_1 the first unwanted cut-off frequency in the passband : $F = f_1 - f_0 = n \cdot f_0$

With v the phase velocity and p the physical length, the electrical length is:

$$\text{at } f_0: \vartheta_0 = \beta \cdot p = \frac{2\pi}{\lambda_0} \cdot p = \frac{2\pi}{v} \cdot f_0 \cdot p$$

$$\text{at } f_1: \vartheta_1 = \frac{2\pi}{\lambda_1} \cdot p = \frac{2\pi}{v} \cdot (n+1) \cdot f_0 \cdot p$$

For $\tan \vartheta_1 = \tan \vartheta_0$ then $\vartheta_1 - \vartheta_0 = k \cdot \pi$

For $k = 1$:

$$n = \frac{v}{2 \cdot p \cdot f_0} \quad \text{and} \quad F = n \cdot f_0 = \frac{v}{2 \cdot p}$$

Fig 1: Calculation of the period.

small or too big, which makes the practical implementation more difficult.

To convert an LC filter into a line filter using the Richards transformation, the length of the stubs are fixed and the impedance levels are plotted against the standardised circuit elements of the LC filter. The calculation is carried out at the cut-off frequency of the filter.

The conversion formulae are listed in Table 1.

2. The Kuroda-Identities

With the help of a unit element (UE), the Kuroda-Identities convert short-circuited

stubs into open circuit stubs. The unit element itself is a section of line with the same length as the stubs and with an impedance of, Z_{UE} , which is yet to be determined. The Identities listed in Table 2 are valid for the circuit elements of low pass filters. The Kuroda-Identities are verified by comparing the iterative matrices of the circuits [3].

3. An example: Chebyshev low pass filter

As an example, let us consider a Chebyshev low pass filter (Fig 2) with the following specification:

- Filter order: $n = 5$

	Standardised	De-standardised	Table 1: Conversion formulae.
Open circuit transmission	$z_L = \frac{Z_L}{R_0} = \frac{l}{\tan \vartheta}$	$Z_L \frac{R_0 \cdot l}{\tan \vartheta}$	
Short circuit transmission	$z_C = \frac{Z_C}{R_0} = \frac{\tan \vartheta}{c}$	$Z_C = \frac{R_0 \cdot \tan \vartheta}{c}$	

$\omega_0 = 2\pi \cdot f_0$ - Cut-off frequency of LC filter
 $\ell = \omega_0 \cdot L / R_0$ - Standardised inductance
 $c = \omega_0 \cdot C \cdot R_0$ - Standardised Capacitance
 R_0 - Characteristic impedance e.g. 50Ω

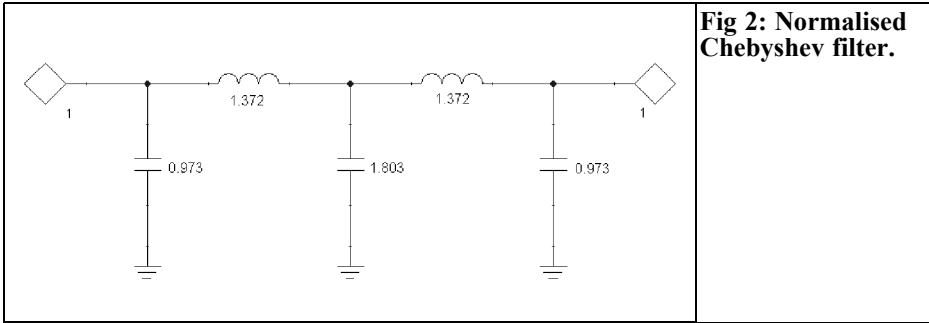


Fig 2: Normalised Chebyshev filter.

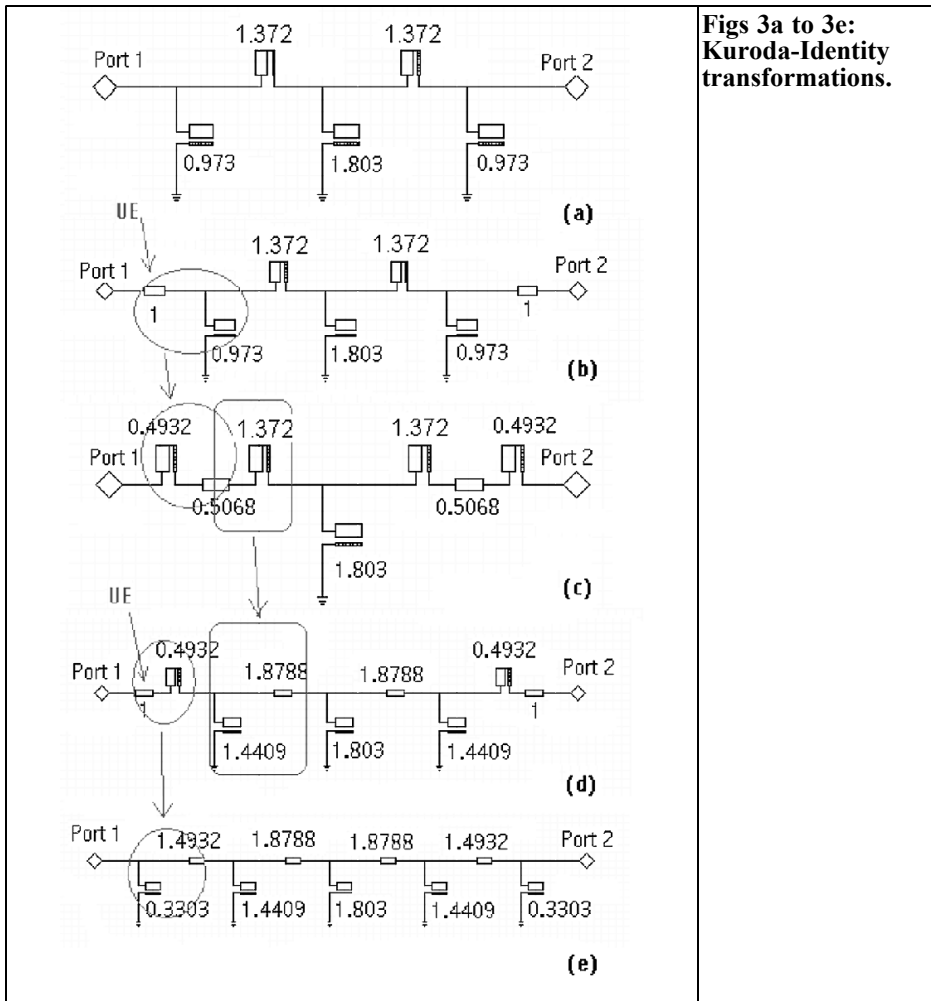
- Impedance level: $R_0 = 50 \Omega$
- f_0 : $2GHz$
- Reflection attenuation: $p \geq 20 dB$

- (or ripple $\alpha \leq 0.04364 dB$).

The starting point for the manual calculation of the line filter is the normalised circuit in Fig 2. The standardised circuit elements can be taken from the ANSOFT Filter Designer (Filter Wizard, User De-

Table 2: Kuroda-Identities for low pass filters.

	A	B
1		
	$z_{UE1} = z_{UE2} + z_L$ $y_C = \frac{z_L}{z_{UE2}(z_{UE2} + z_L)}, z_C = \frac{1}{y_C}$	$z_{UE2} = \frac{z_{UE1}}{1 + z_{UE1} \cdot y_C}$ $z_L = \frac{z_{UE1}^2 \cdot y_C}{1 + z_{UE1} \cdot y_C}$
2		
	$z_{UE1} = \frac{z_{UE2}}{1 + z_{UE2} \cdot y_C}$ $z_L = \frac{z_{UE2}^2 \cdot y_C}{1 + z_{UE2} \cdot y_C}$	$z_{UE2} = z_{UE1} + z_L$ $y_C = \frac{z_L}{z_{UE1}(z_{UE1} + z_L)}, z_C = \frac{1}{y_C}$



Figs 3a to 3e:
Kuroda-Identity
transformations.

fined, Standard G-Values) or the filter catalogue [5]. They are only required in this section for the manual calculation.

The Richards transformation is as simple as can be. For the line length $\lambda/8$ ($\tan\theta=1$), the inductors are replaced by short-circuited lines with $z_L=l$, and the capacitors by open circuit lines with $y_C=c$ (Fig 3a). Unit elements are added from the ports, with the standardised impedance of 1 (Fig 3b). This has no influence on the transmission function of the filter, because these lines are free from reflec-

tions and shorted on both ends. At port 1 then, we have the circuit as per Table 2, B2. It is converted into a circuit as per A2:

$$Z_{UE1} = \frac{1}{1 + 0.973} = 0.5068$$

$$Z_L = \frac{0.973}{1 + 0.973} = 0.4932$$

At port 2, we convert the circuit as per A1 into the circuit as per B1. For reasons of symmetry, the results at port 2 are

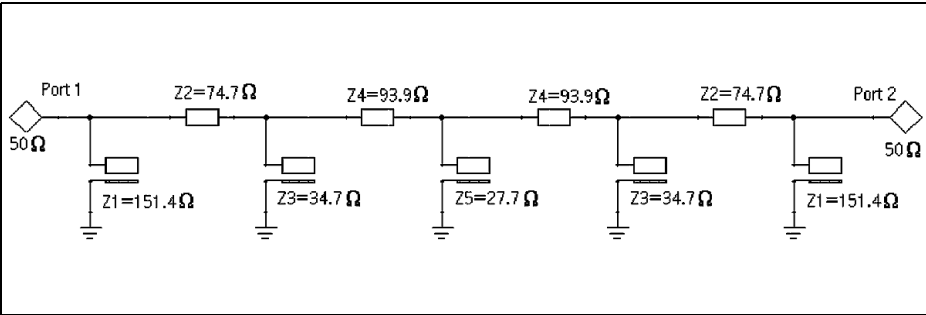


Fig 4: Impedances for filter ($Z_1=1/0.3303 \times 50\Omega=151.40\Omega$; $Z_2=1.4932 \times 50\Omega=74.66\Omega$).

equivalent to those at Port 1. We continue with the conversion in accordance with Fig 3. In the end, the low pass filter consists only of unit elements and open circuit stub cables. De-standardising is carried out as per Table 1 and imped-

ances are indicated for all line sections (Fig 4).

The impedances of the lines are now known. Thus, using whatever tool we select, we can determine the physical

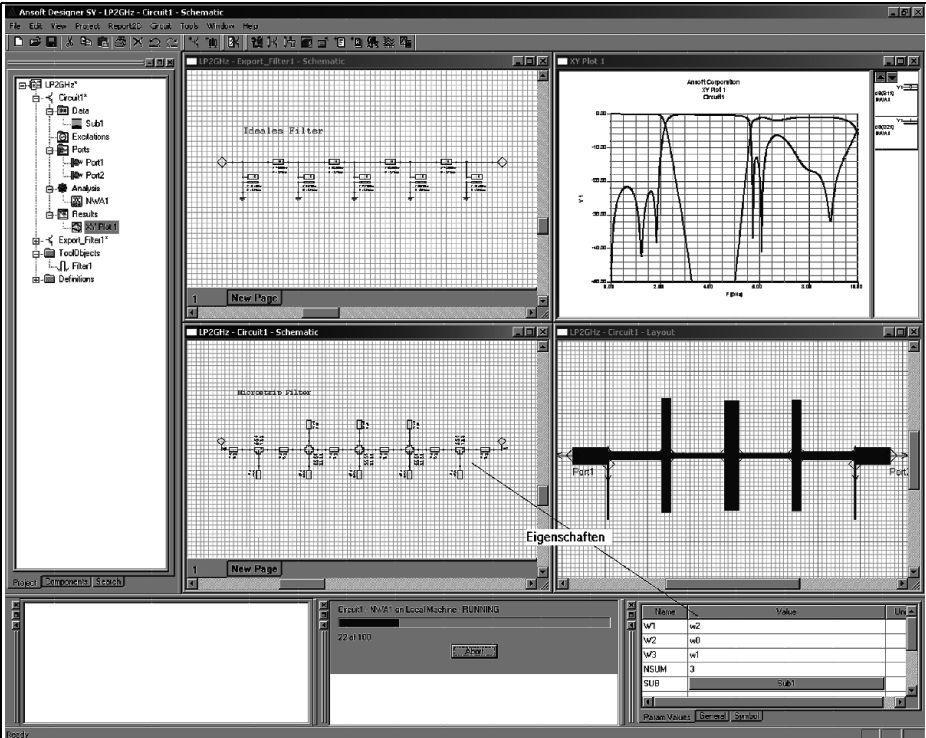


Fig 5: Layout using ANSOFT software.

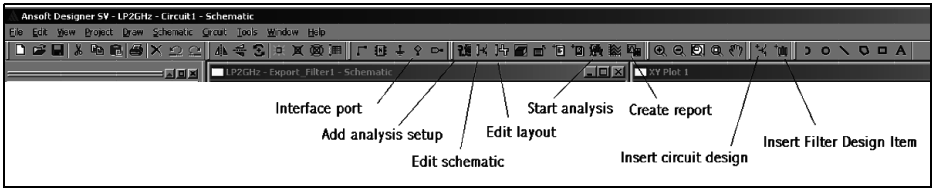


Fig 6: Icons used by ANSOFT software.

dimensions and create the filter from the circuit board material.

4. Filter development using ANSOFT Designer SV

Ansoft Designer [6] is an efficient tool for the analysis and synthesis of linear and non-linear high-frequency circuits and communications systems. Even the free student version is a big software package with lots of good things. It includes the circuit simulator (in the

student version, restricted to linear circuits), a filter design tool, a line calculator, a layout editor and a Smith chart tool. A description of the program's scope, including how it differs from the full version, can be found on the homepage of ANSOFT [6], or in [7].

Note 2: The most recent version of the ANSOFT Designer SV (version 2.2) contains a limited version of the EM simulator.

There are no problems in installing the software. The program can be satisfactorily operated intuitively. Fig 5 shows the layout of the user interface for this project. For the subsequent control of the

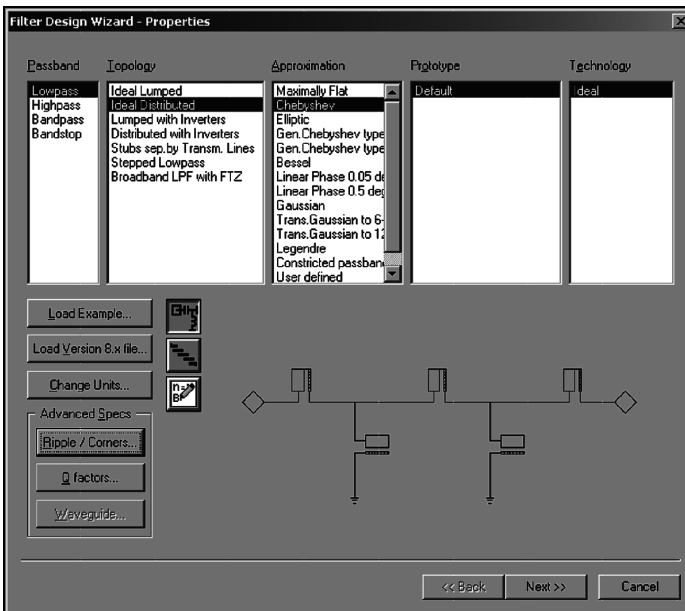


Fig 7: Using the filter design wizard.

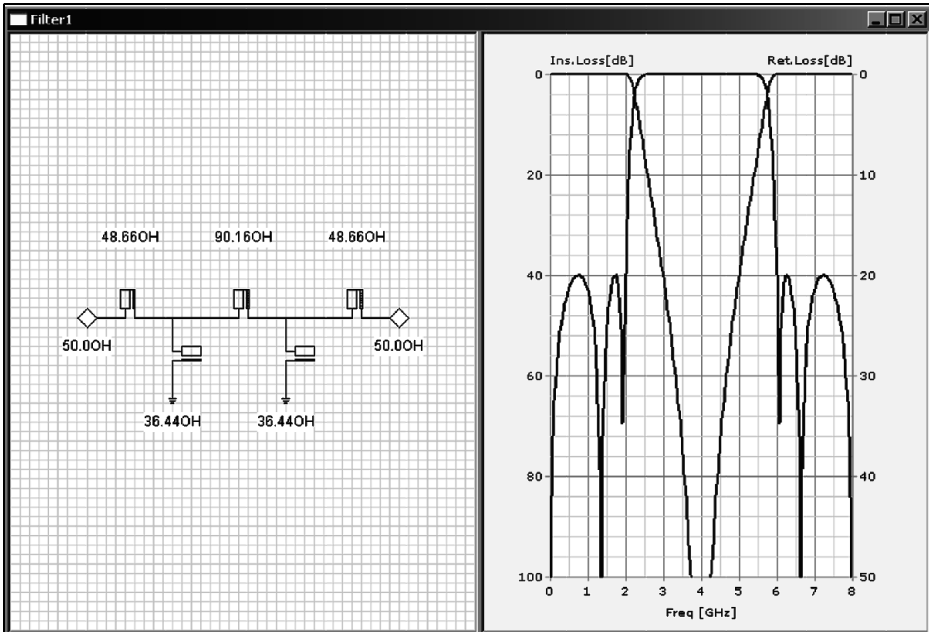


Fig 8: Circuit produced by the filter design wizard.

software, we mainly use the toolbar (Fig 6). Some functions such as the introduction of unit elements can only be used through the menus.

Two different methods of creating the filter are given below. In the first method, we proceed in exactly the same way as for manual calculation. With the ideal filter as a starting point, unit elements are introduced and the Kuroda-Identities are applied. Then the circuit is replaced by stripline components, and we use the “TRL” line calculator to calculate the physical dimensions of the lines from the known line impedances. In the second method, the Filter Wizard automatically replaces the circuit and layout of the filter.

4.1. First method

After starting the program, we immediately call up the “Filter Design Wizard” (Insert Filter Design Item, Fig 7).

As the topology, we select “Ideal Distributed”. For the approximation procedure, we select “Chebyshev”. If we enter “Return Loss = 20dB”, this produces “Ripple / Corners ..” in the dialogue box. The order of filter and the electrical length of the lines can be left as the pre-set values, i.e. 5 and 45°. Enter the frequency (2GHz) and the specification is complete.

We close the specification window and can now consider the circuit in Fig 8. It begins with a series inductance. Right-click on this circuit and on “Dual” in the pop-up menu. We now have the dual circuit, as shown in Fig 3a. The dual circuit is to be preferred because it leads to a symmetrical line filter. Unit elements can be added from the “Filter, Element, Insert, Transmission Line” menu. Left-click (Fig 8) on the stub cable near port 1. This inserts a unit element on the left side of the stub cable. The unit elements must always be inserted starting from the

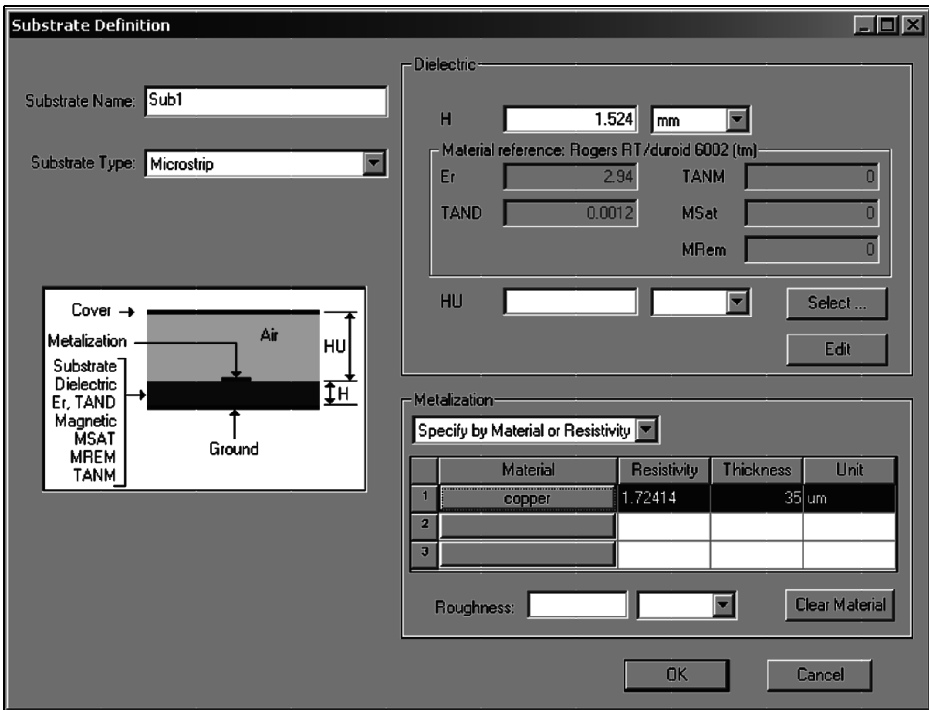


Fig 9: Specification of the PCB material.

ports, since this is the only way to maintain the transmission function of the filter. Since the “Filter Wizard” always inserts the unit elements to the left of a specified “target element”, you have to click on the stub line at port 1 and on the port itself at port 2.

The result should be like Fig 3b, with the difference that the “Filter Wizard” calculates and displays the de-standardised impedances in the same way. Mark the unit element and the stub cable, (left-click, press down and draw) and in the “Filter, Transform” menu click on “Kuroda”. The “Filter Wizard” applies the appropriate Kuroda-Identity to the marked components.

Continue in the same manner, as shown in Fig 3, and finally export (for analysis purposes) the final circuit diagram from

the “Filter, Export, Electrical circuit” menu to ANSOFT Designer. In this menu, you can also decide whether the line lengths and the impedances should be exported as variables. The “Project Manager” applies the “Export_Filter1” directory for the filter. The exported variables can be found (if applicable) under “Circuit, Design Properties, Local Variables”.

4.1.1. Technology / Circuit diagram

In this section, we replace the circuit diagram of the filter using stripline technology. By pressing “Insert circuit design”, we initially set up a page from a drawing. If you want to define the printed circuit board material yourself, close the “Choose Layout Technology” dialogue box by selecting “None”. Make sure that the “Project Manager” has applied a new

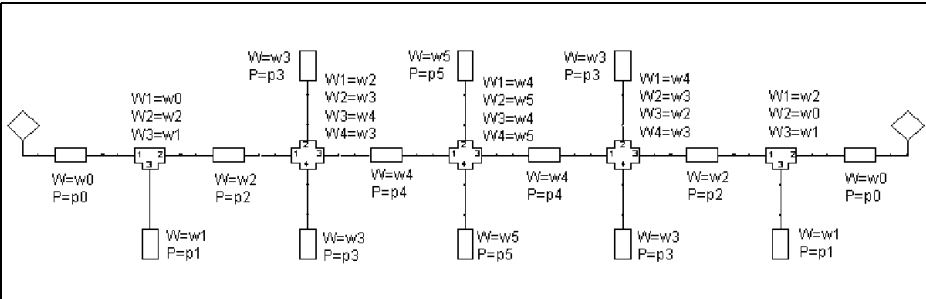


Fig 10: Circuit diagram produced using method 1.

“Circuit1” directory. Open this directory and right-click on ”Data”. In the sub-menu, click on “Add Substrate Definition”. In the “Dielectric” section of the new dialogue box (Fig 9), click on “Select” and then in the new dialogue box “Select Dielectric” click on “Select” again. For the current application, we select Rogers RT/duroid 6002 [8] and close the dialogue box. Enter 1.524mm. as "Height" (material thickness) for the dielectric In the “Metalisation” box (Fig 9), we click under “Material” and select “Copper” for the metalisation, and enter 35µm as the metalisation thickness.

We import two "Interface Ports" from the “Draw” menu (or from the toolbar). In the project manager window, we open the “Components” register menu Then we open the “Microstrip” directory and import one “MS Tee - Ref. Planes at Edge”, and one “MS Cross Junction” from "General Components".

From “Components, Microstrip, Open Ended Line” we import the component “MS Open Stub, Physical Length” and finally from “Components, Microstrip, Transmission Lines” the component “MS Trans. Line, Physical Length”.

Fig 11: Definition of the PCB material for method 2.

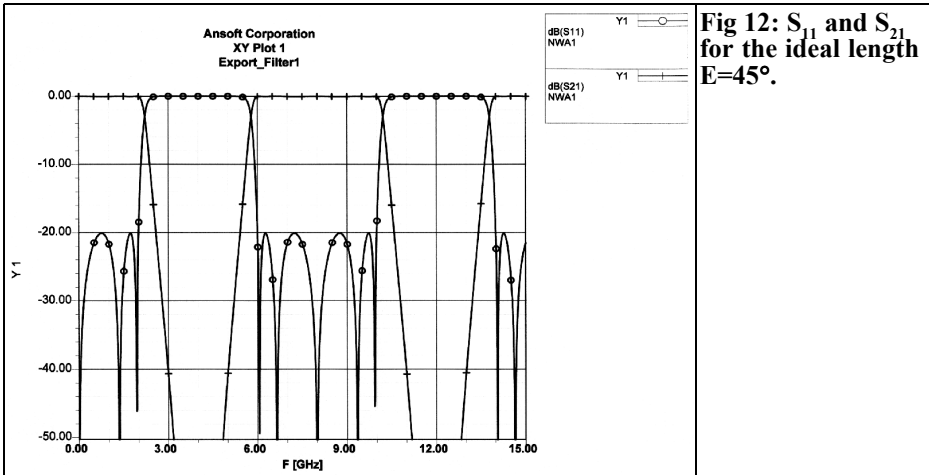


Fig 12: S_{11} and S_{21} for the ideal length $E=45^\circ$.

We now use "Copy/Paste" to complete the circuit diagram from these components (Fig 10). Two stub cables connected in parallel with double the resistance value replace the middle stubs because they have too low a resistance.

4.1.2. Calculation of the lines

The physical dimensions of the lines are calculated from the given resistances using the "TRL" line calculators. The line calculator can be called up either from the "Circuit, TRL" menu or from the "Properties" window of the lines. If

"TRL" is called up from the characteristics window, then the calculated values for the line are assigned when "TRL" is closed. In the right-hand section of the TRL window, enter the line resistance, the desired electrical line length (45°) and the frequency (2GHz). Using "Synthesis", "TRL" calculates the physical width and length of the line.

Unfortunately, the calculator cannot be used for the stubs, so we have to find another way to manage. We position an "accessory line" on the circuit diagram and call up "TRL" to calculate the physical dimensions of stubs with resistances

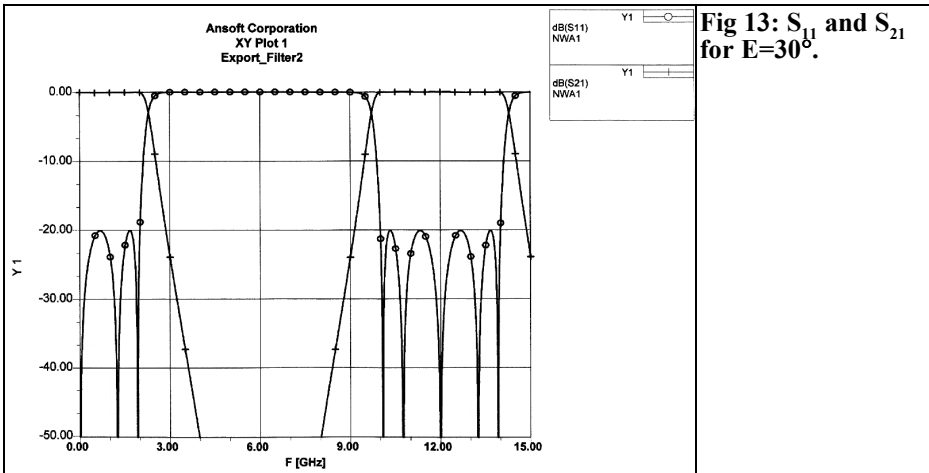


Fig 13: S_{11} and S_{21} for $E=30^\circ$.

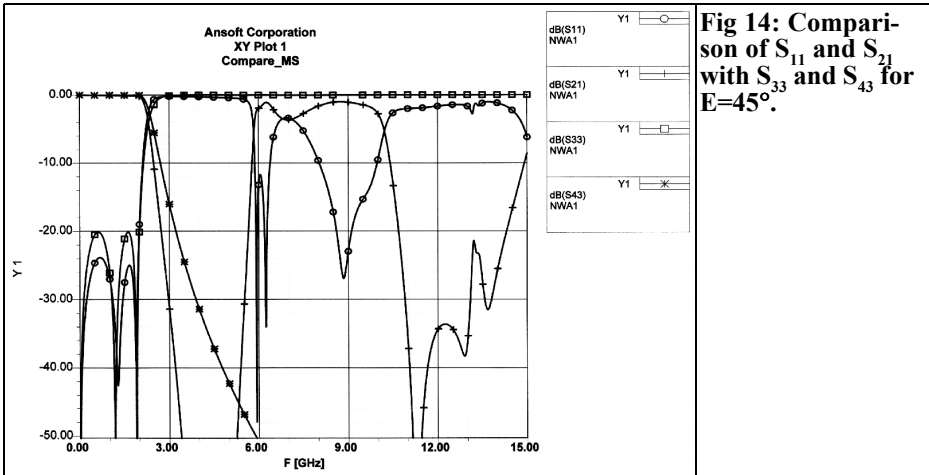


Fig 14: Comparison of S_{11} and S_{21} with S_{33} and S_{43} for $E=45^\circ$.

of 151.4Ω , $2 \times 34.7\Omega = 69.4\Omega$ and $2 \times 27.7\Omega = 55.4\Omega$ and an electrical length of 45° at 2GHz.

We now open the "Details" window by selecting "Details" and note down (on a piece of paper) the effective relative permittivity, K_{eff} ($=\epsilon_{eff}$) for each calculation. We need these values to correct the line lengths. The lengths calculated by "TRL" are too long for open circuit lines. The shortening required, p , can be calculated approximately for each stub with the help of the formula specified in [9]:

$$\frac{\Delta p}{h} = 0.412 \cdot \frac{\epsilon_{eff} + 0.3}{\epsilon_{eff} - 0.258} \cdot \frac{\frac{w}{h} + 0.262}{\frac{w}{h} + 0.813}$$

where:

- h = height of substrate
- w = width of line
- ϵ_{eff} = effective relative permittivity.

The complete circuit diagram and the final physical dimensions of the lines are shown in Fig 10.

4.1.3. Layout of filter

The layout editor is called up by selecting "Edit Layout". ANSOFT Designer should already have set up the filter

layout! If necessary, align the components by marking them and clicking on "Align Microwave Ports". Unfortunately, layouts cannot be exported using the student version.

4.2. Second method

4.2.1. Specification and setting up of filter

Start the Filter Wizard again and this time select "Stubs sep. by Trans. Lines" as the topology and "Microstrip" as the technology. All other entries remain unchanged from the first method. In the following dialogue box "Medium Properties" (Fig 11), we enter the known parameters of the substrate. If the Filter Wizard displays no errors, the filter in question is completely ready and it can be exported to "Ansoft Designer" for analysis purposes. In the "Filter, Export" menu, we now select "Physical circuit". In this case too, the "Project Manager" sets up a directory - for example, "Export_Filter2" - for the filter. Double left-click on this directory, which opens the circuit diagram editor.

4.3. Analysis of filter

To simulate the filter, you must first define the frequency range. If you have



used the first method, you must enter the start and end frequencies in the dialogue box “Add analysis setup” yourself, together with the step sizes. If the second method is used, the frequency range and the step sizes are automatically pre-selected by Filter Wizard. The analysis is begun using “Start analysis”. When the analysis has been carried out, use “Create Report” to call up the dialogue box of the same name. The representation can be in the form of a Smith diagram, polar or Cartesian co-ordinates, or a table. In the following dialogue box, “Traces”, we specify the variables that should be displayed. Apart from the variables that are normally on offer, we can define our own variables (Output Variables) and thus utilise the mathematical functions of the Ansoft Designer.

The reflection coefficient, S_{11} , and the transmission coefficient, S_{21} , of the filter are of interest here. These are added to the report via “Add Trace”. “Done” closes the dialogue box and displays the behaviour of S_{11} and S_{21} over the frequency. (Figs 12 to 14). For the filter in Fig 13, the electrical line length is 30° . The line impedances lie in the range between 32Ω and 138Ω and are still relatively easy to obtain. The pass bands have been pushed further apart than for the electrical line length of 45° in Fig 12. A comparison with the ideal LC low pass with the same specification can be seen in Fig 14.

5. Summary

The Richards transformation and the Kuroda-Identities are briefly explained, together with the derivation of a line low pass filter from an LC low pass. The filter is developed in two different ways, using ANSOFT Designer, and some characteristics of the software are shown

in the process. With the help of the Richards transformation, low pass and high pass filters are developed. The periodicity of the passband, which is unavoidable for line filters anyway, is used to derive band stops and pass band from a low pass or a high pass filters [4].

6. Literature

[1] P. I. Richards, Resistor-Transmission-Line Circuits, Proc. IRE, Febr. 1948, pp. 217-220

[2] Zinke-Brunswig, High-frequenc Engineering 1, Springer, 6th edition, 2000

[3] R. Ludwig, P. Bretchko, RF Circuit Design, Theory and Applications, Prentice-Hall, 2000

[4] Max W. Medley, Microwave and RF Circuits: Analysis, Synthesis and Design, Artech House 1993

[5] Rudolf Saal, Filter Design Manual, Hüthig Verlag, 1988

[6] www.ansoft.com bzw. www.ansoft.com/ansoftdesignersv/capabilities.cfm

[7] David Vye, Free Software is powerful Simulator, Microwaves & RF, Aug. 2003 (www.ansoft.com/news/articles/03.08_MWRF.pdf)

[8] www.rogerscorporation.com

[9] E. O. Hammerstad, F. Bekkadal, Microstrip Handbook ELAB Report STF44 A74169, The Uni. of Trondheim, 1975, P. 25



Alexander Meier, DG6RBP

ATV transmitter with PLL for 10GHz

The ATV mode (amateur radio TV) is becoming increasingly popular, this is because many ATV repeaters have become active in recent years. This has meant that the 3cm band is becoming more and more important, in addition to links, it is increasingly used for operator input and output.

With this in mind, a modern, frequency stabilised ATV transmitter has been developed, the transmit frequency can be simply and accurately selected at any point on the entire 10GHz amateur radio band. This makes it suitable for both use as a transmitter in a repeater or for direct operation.

1.

Introduction

A transmitter in an ATV repeater, e.g. as operator output or for a link, is subjected to temperature variations and is in operation for long periods of time. So the transmit frequency should change as little as possible. For direct operation, on the other hand, simple frequency adjustment is important, particularly if it is used to work with different repeaters. The PLL stabilised transmitter has a few advan-

tages here: the transmit frequency can be selected simply and easily anywhere in the 3cm. band, by means of a micro-controller, and frequency deviations are prevented by the PLL!

The transmitter with a Phase Locked Loop (PLL) is based on Voltage Controlled Oscillators (VCOs), that have recently become available at reasonable prices up to and beyond 10GHz [1]. Though you won't find a suitable VCO that will cover the entire 3cm band with an acceptable tuning rate. But there is an option to use a VCO of 5.00 to 5.25GHz then doubling and amplifying the output signal. There is, however, a disadvantage of this approach; compared with free running oscillator or DRO transmitters, the circuit complexity is considerably greater! Fig 1 shows the block diagram of the project implemented here. A VCO of 5.00 to 5.25GHz is stabilised using a PLL and after a passive doubler (x2) covers the entire 3cm band from 10.00 to 10.50GHz. The losses that this generates are balanced out again using two amplifier stages with an output of approximately 1mW at 10GHz. This is sufficient to drive an additional power amplifier, e.g. a small TWT or a 200mW FET amplifier.

The PLL is controlled by means of an 8051 micro-controller from ATMEL. The transmit frequency can thus be con-

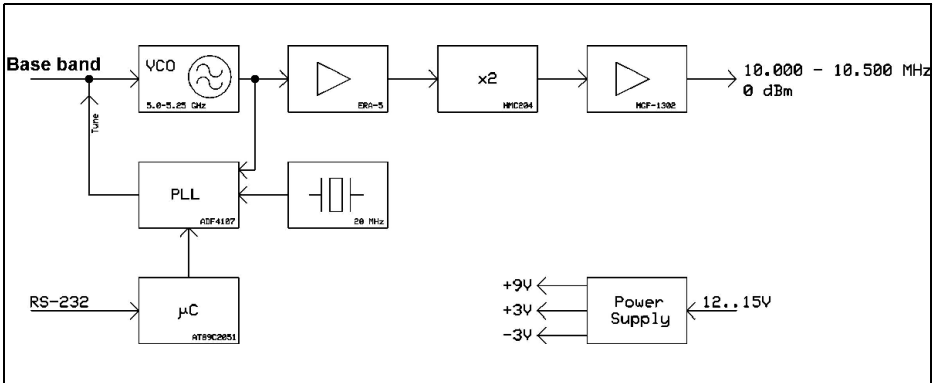


Fig 1: Block diagram of the ATV transmitter.

veniently selected anywhere on the 3cm band via the RS-232 interface from a PC or through a separate control module. It is not obligatory to use the interface for fixed frequency operation in a repeater because the transmitter starts at a fixed frequency when it is switched on (pre-programmed in the micro-controller).

232 interface.

2.

Circuit description

An LED signals when the PLL is locked. If this is not the case, the micro-controller switches the amplifier stages off and sends an error string via the RS-232 interface.

As an option, the ATV transmitter can also monitor an externally connected power amplifier (PA); if the voltage at the monitor output of the external PA falls below a selected threshold value, the micro-controller reports this via the RS-

Fig 3 shows the circuit diagram of the transmitter and Table 1 shows the parts list. The VCO, U2 can be tuned at pin 22, using a tuning voltage of between 0 and 10V giving between 5.0 and 5.5GHz at Pin 16. Since only a frequency range of 5.0 to 5.25GHz is required before doubling, a tuning voltage of between 0 and 5V is satisfactory.

The operating voltage is fed to the VCO via a low pass filter (L2 and C15-C17) in

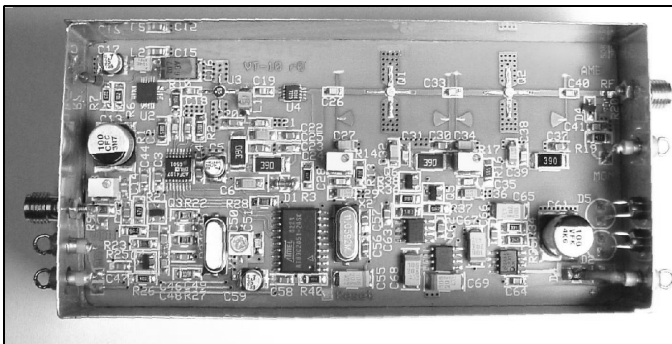


Fig 2: Picture of the completed ATV transmitter.

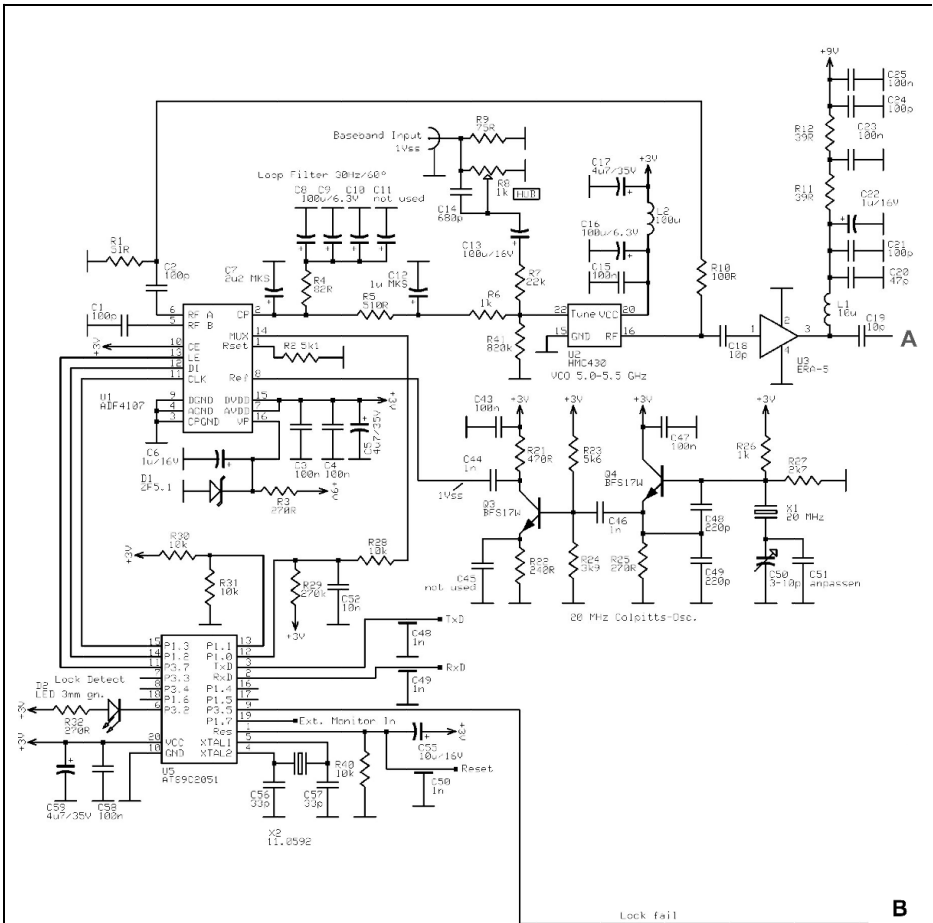


Fig 3: Circuit diagram of ATV transmitter.

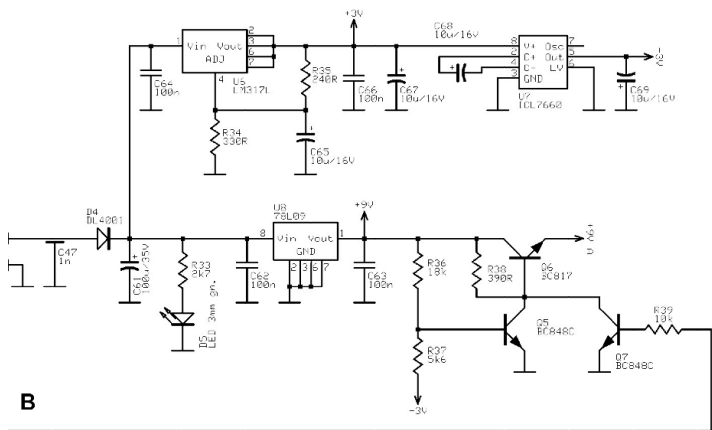
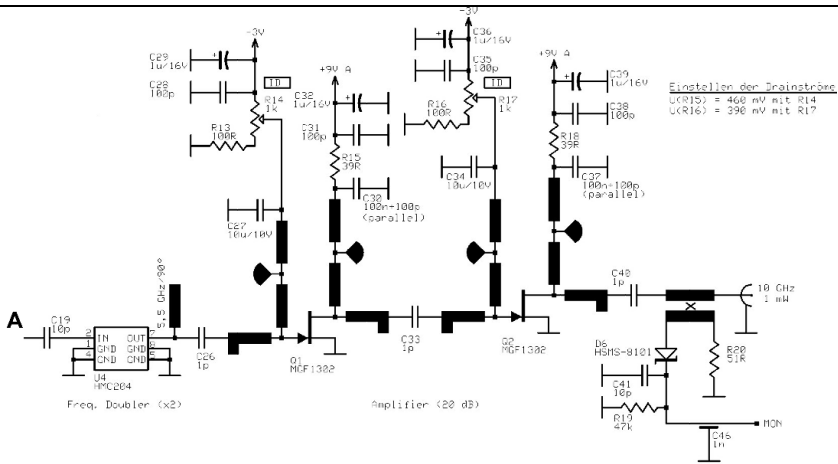
order to suppress noise and other residual frequencies in the supply voltage.

The VCO's output is approximately 1mW, one part of this is fed to the PLL via the resistor, R10. The majority goes to the amplifier, U3. This gain block increases the power to a few mW to feed the passive doubler, U4. A stub at the output of the doubler suppresses the first harmonic at 5GHz. The second harmonic at 10GHz (approximately -20dBm) corresponds to the desired frequency and is amplified, using the Mitsubishi FETs Q1

and Q2, to an output of approximately 0dBm (1mW). The quiescent currents of the amplifier are set by means of the trimmer potentiometers, R14 and R17.

If required, the printed circuit board provides the option of a monitor circuit with a directional coupler and a Schottky diode (R19, R20, C41, C46, D4).

The PLL module (U1) from Analog Devices has the output frequency of the VCO (5.00 - 5.25GHz) fed into it via pin 6, and the reference frequency (20MHz)



via pin 8. The phase comparison is carried out at 100kHz, so that there is a more than satisfactorily small step size of 0.2MHz, however, only frequency steps of 1MHz are supported by the microcontroller software. This is more than sufficient for ATV purposes. The loop filter of the PLL (C7, R4, C8 to C10, R5, C12) has been designed for the transmission of video signals. The loop bandwidth is 30Hz. The low frequency of the phase comparison makes it possible to use practical component values. High-quality foil capacitors must be used for

C7 and C12, and good tantalum capacitors for C8 to C10.

The tuning voltage is combined with the modulation signal (base band) via R6 and R7 before the VCO. The trimmer, R8, serves to set the frequency deviation. The capacitor, C14, is used to provide for a slight increase in the high modulation frequencies.

A simple quartz oscillator in a Colpitts circuit (Q4) is used as a reference oscillator. The frequency can be raised some-

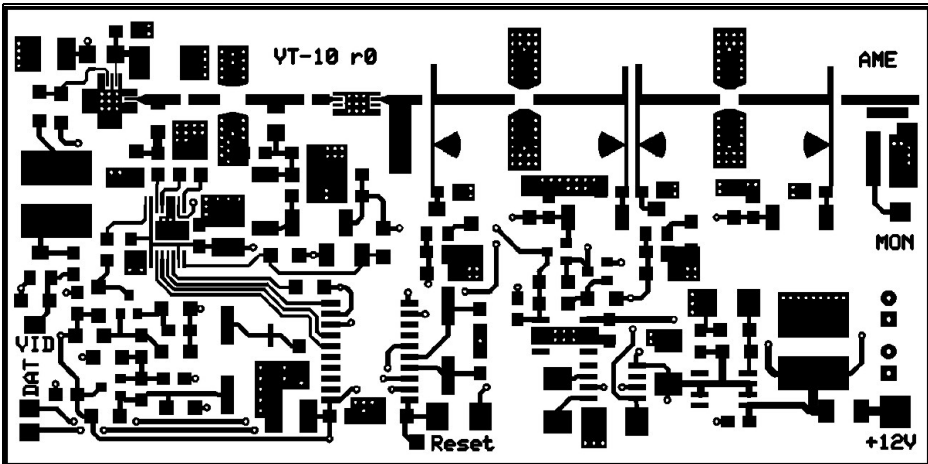


Fig 4: Top side PCB layout for the ATV transmitter.

what using the trimmer capacitor, C50. A subsequent amplifier stage (Q3) serves mainly as a buffer. If required, the capacitor, C45, could be used to increase the AC voltage amplification of the stage.

The PLL's programming is carried out by an 8051 micro-controller (U5) from AT-MEL via pins 11, 14 and 15. The

controller operates with a quartz crystal frequency of 11.0592MHz to generate a baud rate of 9600 Baud. The lock-detect signal of the PLL is compared with a pre-set voltage, using an integrated comparator (Pin 12, 13). If the current at pin 12 exceeds the reference voltage of 2.5V, the controller recognises that the PLL has locked and switches the amplifier stages on via pin 9. In addition, an LED signals to pin 6 that the PLL has locked.

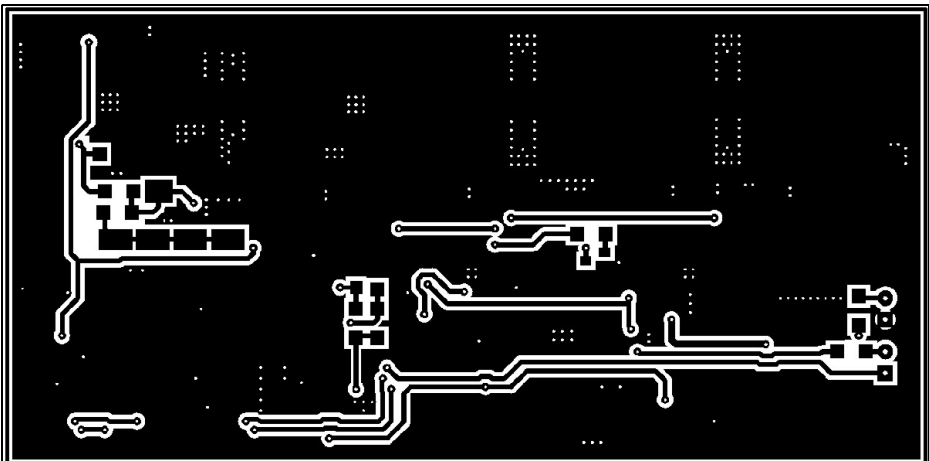


Fig 5: Bottom side PCB layout for the ATV transmitter.

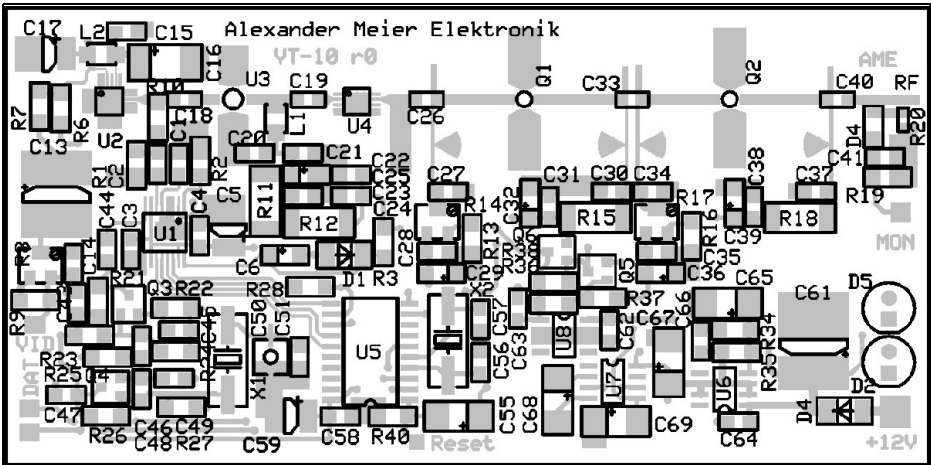


Fig 6: Top side component layout for the ATV transmitter.

For the optional supervision of an external output amplifier stage, the corresponding monitor signal is compared using a comparator (external circuit as and when required). The output signal of the comparator is then fed to the micro-controller via pin 19. A voltage level of 0V means the PA has failed, while a level of 5V means it is operating. The status of pin 19 is continuously monitored in the controller, and should any

change occur a message is transmitted via the RS-232 interface.

The supply voltage for the module (12 to 15 Volts) is fed in through a reverse battery protection diode (D4). The fixed voltage regulator (U8) uses it to generate the operating voltage for the amplifier stages (9V). This is applied, through the protective circuit of Q5-Q7, only if the

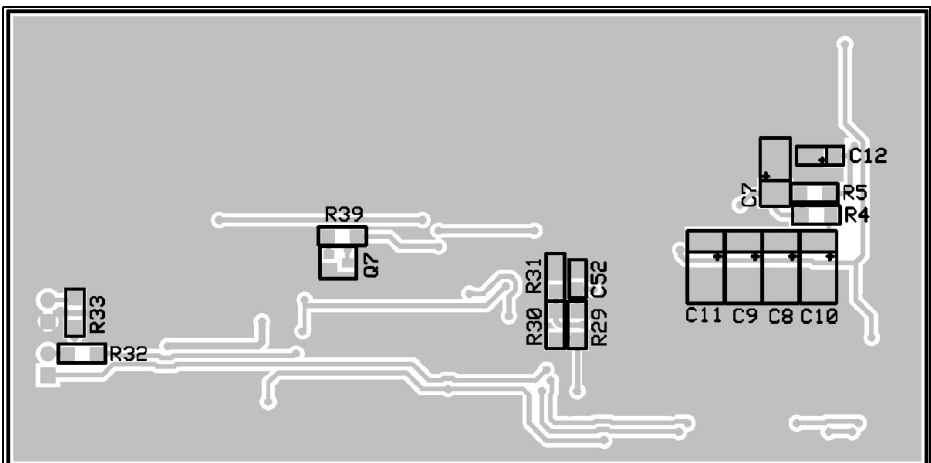


Fig 7: Bottom side component layout of the ATV transmitter.

**Table 1: Parts list for the ATV transmitter.**

R1, R20	51R	1206		C27, C34	10 μ F/10V SMD Tantalum
R9	75R	1206		C55, C65, C67-C69	10 μ F/16V SMD Tantalum
R4	82R	1206		C8-C10, C16	100 μ F/6.3V SMD Tantalum
R10, R13, R16	100R	1206		C13	100 μ F/16V SMD Electrolytic capacitor
R22, R35	240R	1206		C61	100 μ F/35V SMD Electrolytic capacitor
R3, R25, R32	270R	1206		C50	C-Trimмер 3 - 10pF (SMD)
R34	330R	1206		C46-C50	Feedthrough capacitor 1nF
R38	390R	1206		L1	10 μ H SIMID 1210
R21	470R	1206		L2	100 μ H, SIMID 1210
R5	510R	1206		D1	ZF5.1
R6, R26	1k	1206		D4	DL4001
R27, R33	2k7	1206		D2, D5	LED 3mm green low current
R24	3k9	1206		D6	HSMS-8101
R2	5k1	1206		Q1, Q2	MGF 1302
R23, R37	5k6	1206		Q3, Q4	BFS 17W
R28, R30, R31, R39, R40	10k	1206		Q5, Q7	BC 848C
R36	18k	1206		Q6	BC 817
R7	22k	1206		X1	20MHz, SMD C _L =16pF
R19	47k	1206		X2	11.0592MHz, SMD
R19	270k	1206		U1	ADF4107 BRU
R41	820k	1/4W		U2	HMC 430 LP4
R11, R12, R15, R18	39R	2510		U3	ERA-5
R8, R14, R17	1k	SMD	Trimmer	U4	HMC 204 MS8G
C26, C33, C40	1pF	0805		U5	AT 89C2051 (SMD), programmed
C18, C19, C41	10pF	0805		U6	LM 317L (SMD)
C51	18pF	0805		U7	ICL 7660 (SMD)
C56, C57	33pF	0805		U8	78L05 (SMD)
C20	47pF	0805			1 x tinplate housing 111 x 55 x 30mm
C1, C2, C21, C24, C28, C30, C31, C35, C37, C38	100pF	0805			1 x LP DG 6 RBP-VT 10
C48, C49	220pF	0805			2 x SMA socket
C14	680pF	0805			
C44, C46	1nF	0805			
C52	10nF	0805			
C3, C4, C15, C23, C25, C30, C37, C43, C47, C58, C62, C63, C64, C66	100nF	0805			
C6, C22, C29, C32, C36, C39	1 μ F/16V SMD				
	Tantalum				
C12	1 μ F	MKS-2			
C7	2.2 μ F	MKS-2			
C5, C17, C59	4.7 μ F/35V SMD				
	Electrolytic capacitor				

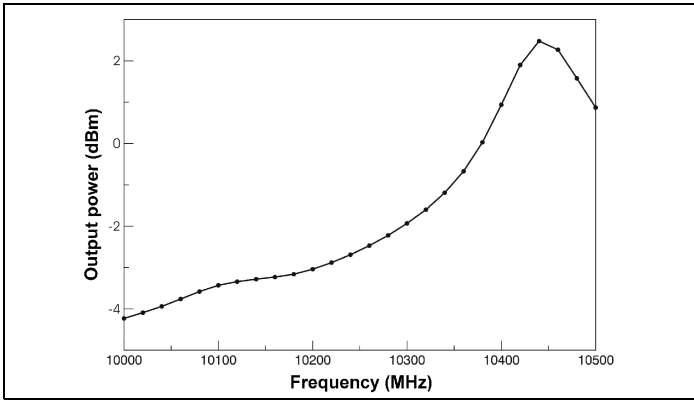


Fig 8: Graph of output power against frequency for the ATV transmitter.

PLL is locked and the negative gate voltage is correct.

An easily adjustable voltage regulator, type LM 317 (U6) is used for the power supply for both the PLL and the VCO (3V). The charge pump (U7) uses this to generate a negative voltage for the gates of the amplifier stages.

3. Assembly and tuning

The entire circuit is built on a 109 mm x

54 mm x 0.51 mm printed circuit board. The inexpensive RO-40003C from Rogers is used as the printed circuit board material. To make assembly easier, particularly for the VCO (U2) and the doubler (U4), the printed circuit board is through-hole plated and coated with a solder stop varnish.

After a some amount of solder paste has been applied, the VCO is melted on, using a reflow soldering unit or a hot air station. This component cannot be mounted using a soldering iron because of the VCO package connections are on the underside of VCO! This is the same for the doubler (U4), it has an earth side on its underside, which must be soldered correctly.

Table 2: Programming control by the RS-232 port on the ATV transmitter.

Controller to ATV transmitter	F10xxx	New transmit frequency 10xxx = frequency in MHz
Controller to ATV transmitter	S	Request transmitter status
ATV transmitter to Controller	P	Power on
ATV transmitter to Controller	L	PLL locked
ATV transmitter to Controller	U	PLL un-locked
ATV transmitter to Controller	A0	External monitor signal fail
ATV transmitter to Controller	A1	External monitor signal OK
ATV transmitter to Controller	E	Error
ATV transmitter to Controller	ERR	Error
ATV transmitter to Controller	OK	New frequency confirmed
ATV transmitter to Controller	F10xxxyAz	Transmitter status 10xxx = frequency in MHz y=L,U (locked, un-locked) z=0,1 (external monitor 0,1)

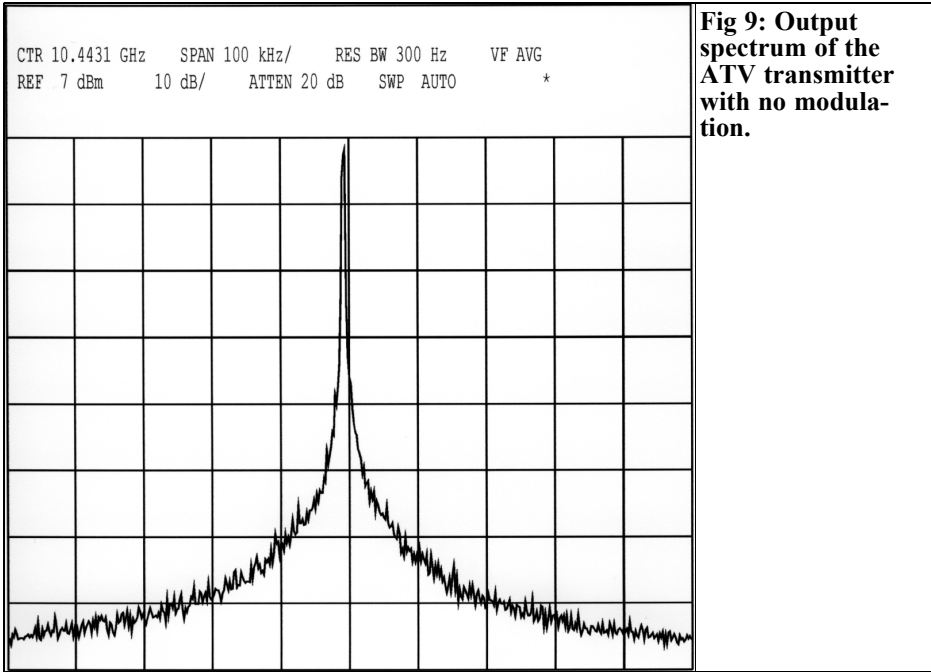


Fig 9: Output spectrum of the ATV transmitter with no modulation.

Finally, all other components apart from Q6 are mounted, with no particular instructions. A soldering iron can be used here. Good eyes and steady hands are required for the PLL (U1) when the components are mounted by hand. The wired resistor, R41 is soldered in “quick as a flash” by hand, depending on the earth (e.g. housing wall). The PCB layout

is shown in Figs 4 and 5 and the component layout in Figs 6 and 7.

Prepare the tinfoil housing by drilling the appropriate holes. Clean the flux residue off of the printed circuit board and fit it into the housing soldering it all around the underside. Then the printed circuit board is cleaned again.

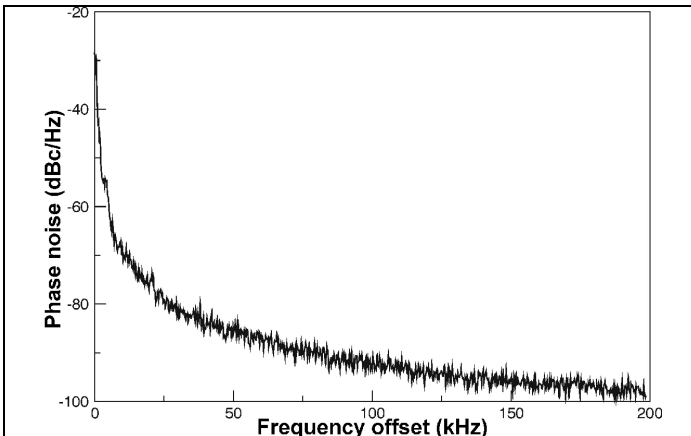


Fig 10: Phase noise performance of the ATV transmitter.



Once the printed circuit board has been connected to the feedthrough capacitors, the first function test can be carried out. When the supply voltage has been applied, the voltage on the gates of, Q1 and Q2, is initially set to -3V, using the trimmers, R14 and R17. Switch off and fit transistor Q6. Switch on again and check the important parts of the circuit. The most important elements are the operating voltages (3V, -3V, 9V, 5.1V at pin 16 of U1) and the 20MHz quartz oscillator. Next the tuning begins.

When the supply voltage is applied, the module starts at the default, pre-programmed, frequency. Now the quiescent currents of the amplifier stages are set. First, the voltage is measured across the resistor R15 and adjusted to 460mV using the trimmer R14. The voltage across R18 is set to 390mV using R17. A spectrum analyser is used to check whether the PLL has locked correctly. The output will still be somewhat below 1mW. Finally, the module is connected to a PC via the RS-232 interface and the frequency is altered from 10.000 to 10.500GHz in steps of 50MHz and check that the PLL has locked correctly. The stubs at the input and output of the transistors, Q1 and Q2, are shortened slightly with a scalpel to find the maximum output in the middle of the frequency range or at any preferred frequency. The frequency deviation of the transmitter is set to $\pm 3.5\text{MHz}$ (as per DL2CH ATV standard [2]) using a spectrum analyser and the carrier minimum method. A 1.5MHz signal is applied at the base band input, with an amplitude of $1V_{pp}$. Increase the shift until the first time that the carrier frequency reaches a minimum. The frequency deviation can also be set to a higher value if required.

After a final function test with a test image and the transmitter is ready for operation.

Table 3: Technical data for ATV transmitter.

Frequency range	10.000 to 10.500GHz
Steps	1MHz
Frequency stabilisation	PLL
Long term stability	$\pm 5\text{ppm}$ / year
Temperature stability	$\pm 50\text{ppm}$ (+10 - +40°C)
Modulation input	Baseband $1V_{pp}$ (positive input gives positive modulation)
Modulation type	FM
Modulation depth	$\pm 3.5\text{MHz}$ for $1V_{pp}$ (intensely adjusted with reserve)
Modulation freq. range	30Hz to >7MHz
Interface	RS-232, 0/3V signalling, 9600 baud, 8 data bits, 1 stop bit, no parity
Interface cable length	20m
RF output impedance	50Ω
Typical output	1mW / 0dBm ($\pm 3\text{dB}$)
Typical harmonic output	<-40dBc (5 - 5.25GHz) <-40dBc (20 - 21GHz)
Other harmonics	<-50dBc
Supply voltage	12 to 15V DC
Supply current	approximately 160mA
Temperature range	+10 to +40°C
Connectors	SMA socket for RF output and baseband input Feedthrough capacitors for RS-232
Indicators	Power - green LED Locked - green LED
Case	Tinplate housing
Size	111 x 55 x 30mm approx.

4.

Programming

The programming of the default frequency of the module, i.e. the frequency at which the module starts when the supply voltage is applied, is pre-set once and for all in the software of the micro-controller (U5). If the transmitter were used in an ATV repeater, this would be



the output frequency. The transmitter could then be used without utilising the RS-232 interface.

Otherwise, the output frequency can be changed at any time via the RS-232 interface in steps of 1MHz. The PLL would allow for smaller steps, but these are not expedient for ATV applications. To set the frequency, we can use a PC with a USB programming adapter [3], another micro-controller with an RS-232 interface, or a frequency input module developed individually for the transmitter with 7-segment displays, described in the next article. Direct connection to the RS-232 interface of a PC is possible, using a level converter (0/3V, e.g. MAX232). If the transmitter is connected to a controller with a high resistance pull-up on the serial communications pins, e.g. like the 89C52, also a 0/5V level could be used.

Table 2 lists the interface commands. Thus, when the supply voltage is applied, the transmitter transmits a "P" for "Power On". As soon as the PLL is locked, the transmitter reports this with an "L". If we now wish, for example, to set the frequency to 10.450MHz, we transmit the character string "F10450" to the transmitter. The new frequency is confirmed and selected using "Ok". As the PLL is not locked during the frequency change, the transmitter reports this with "U". As soon as the PLL is locked, we receive an "L" as a response.

We can check the transmitter status by sending the character "S" to the transmitter. For example, if the transmitter answers "F10200LA1", this means: frequency 10200MHz, PLL locked (L) and monitor signal for external amplifier stage OK (A1).

5.

Readings

The technical data of the ATV transmitter are summarised in Table 3. As the transmitter modulates the output frequency towards higher frequencies for positive voltages on the base band input, the video signal must be inverted before being applied to the base band input to obtain the inverted image usually required for ATV repeater outputs.

The output power at different frequencies can be seen in Fig 8 The frequency of the maximum power is determined during tuning, tuning was carried out here at 10440GHz.

Fig 9 shows the output spectrum without modulation and the phase noise of the transmitter is shown in Fig 10.

6.

References

- [1] Hittite Microwave Corporation, 20 Alpha Rd., Chelmsford, MA01824, www.hittite.com
- [2] FM ATV standard for IARU region 1, Lillehammer conference report 1999
- [3] Convenient procedure for filling VX-2 memories, CQ-DL 10/04, DARC-Verlag, Baunatal, 2004



Alexander Meier, DG6RBP

Frequency input module for 10GHz ATV transmitter module

The previous article describes a 10GHz ATV transmitter module with a serial interface. For normal station operation it would be nice to have a simple option for adjusting the transmitter frequency. A frequency input module was developed for this purpose, with a 5 digit frequency display and three station keys for preferred frequencies. One LED displays the status of the transmitter module's PLL, while another shows that of an externally monitored output stage.

1.

Introduction

To operate the 10GHz ATV transmitter module in an ATV repeater, the transmit frequency is pre-selected and fixed in the module by the micro-controller, U5. Once the operating voltage has been applied, the module then immediately transmits at the correct output frequency. The frequency can be set via the serial interface using a suitable adapter or a PC

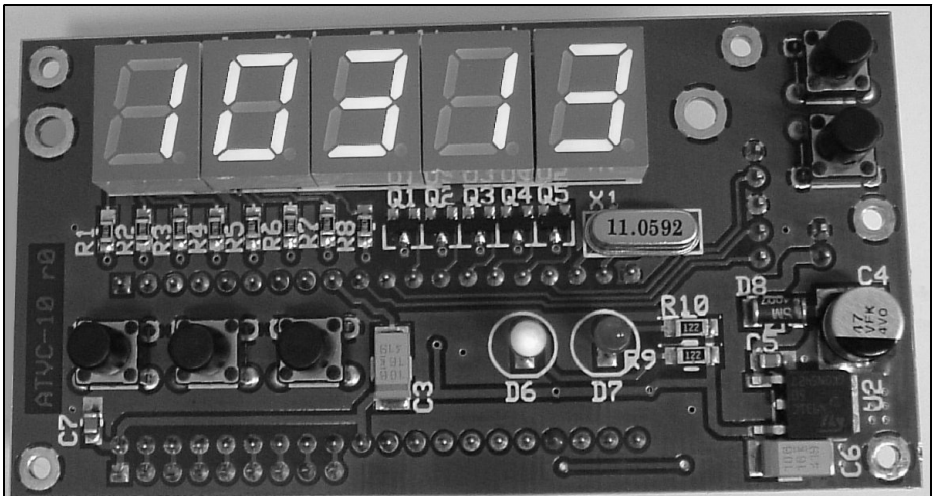


Fig 1: Photograph of the input module for the ATV transmitter.

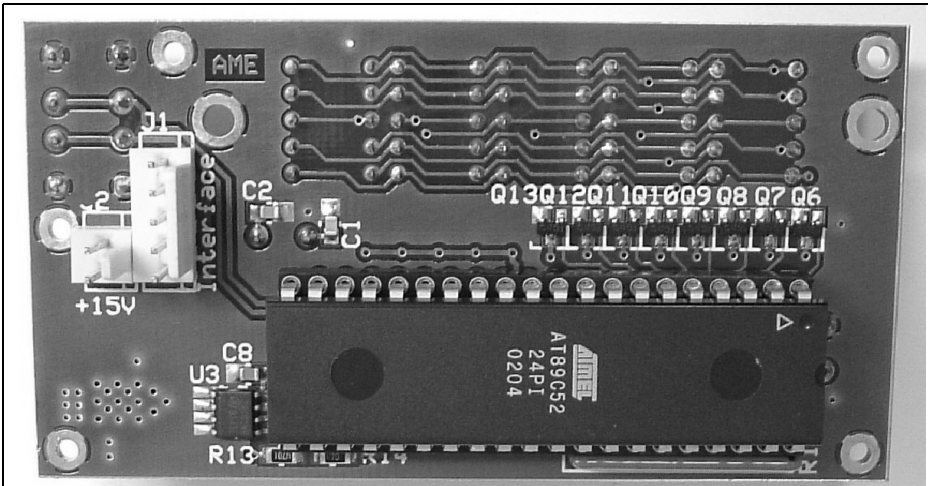


Fig 3: Photograph of the input module for the ATV transmitter, showing the micro-controller.

but this is a very laborious task so a frequency input module has been developed.

The frequency input module is linked to the 10GHz ATV transmitter module via the serial interface. The current transmit frequency is shown on five 7 segment displays. The display can be read easily, even in the dark. The frequency can be adjusted very conveniently, using two keys (Up/Down), in steps of 1MHz. In addition, three preferred frequencies, e.g. of the nearby repeaters, can be stored in a non-volatile memory, so that they can be easily called up using station keys. Figs 1 and 3 show the module.

2.

Circuit description

Fig 2 shows the circuit diagram of the frequency input module and Table 1 shows the parts list. The control is handled by a micro-controller from Atmel, the AT89C52 (U1). The software is stored in the controller's flash memory. Once

Table 1: Parts list for input module for ATV transmitter.

R1-R8	47R	0805
R9, R10	1k2	1206
R13, R14	4k7	0805
R12	10k	1206
R11	8x10k	
C1, C2	33p	0805
C5, C7, C8	100n	0805
C3, C6	10m/16V	Tantalum
C4	47m/35V	SMD Electrolytic capacitor
U1	AT89C52	DIP40, programmed U2
L4931CDT50	U3	AT24C01A
D1-D5	7-segment display, 10 mm, common anode, SA39-11 green	
D6, D7	LED red, 3mm, low current	
D8	DL4001	
Q1-Q13	BCV47	
X1	11.0592MHz, HC49/U-S	
J1	Board connector, 5-pin	
J2	Board connector, 2-pin	
B1-B5	Key, APEM DTS-64N	
5 x	Key caps, round, black	
1 x	IC base, 40-pin	
1 x	Printed circuit board	

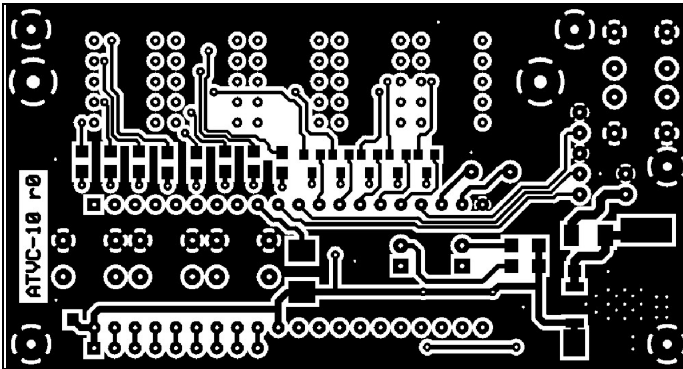


Fig 4: Top view of PCB for input module for ATV transmitter.

the operating voltage has been applied, the controller is reset, using the capacitor, C3. All data such as the content of the station keys, are stored in EEPROM U3. The 7 segment displays, D1 to D5, are driven in time multiplex by the microcontroller. The individual displays are activated one after another with 1ms multiplexing.

The link between the frequency input module and the ATV transmitter goes through the RS-232 interface on the connector, J1. To achieve this, the connections RxD and TxD are crossed - i.e. the RxD connection on the frequency input module is linked to TxD on the transmitter module.

The supply voltage of the module can vary between 9 and 16V, and is supplied through the reverse battery protection diode D8 to the 5V voltage regulator, U2.

The entire circuit fits onto a double sided board measuring only 85 x 45 mm. There

are no unusual factors involved in mounting the components. The PCB layout is shown in Figs 4 and 5 with the component layout in Figs 6 and 7.

3.

Operation

When the supply voltage is applied to both the ATV transmitter and frequency input modules, the transmitter starts at its internally stored transmit frequency. Within less than 2 seconds, the frequency input module then takes over control and sets the transmit frequency to the stored frequency, M1.

The transmit frequency can now be set anywhere between 10.000 and 10.500GHz, using the Up/Down keys, in steps of 1MHz. When a key is held

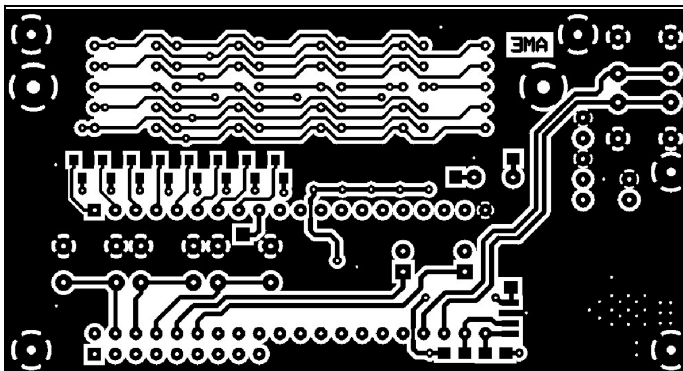


Fig 5: Bottom view of PCB for input module for ATV transmitter.

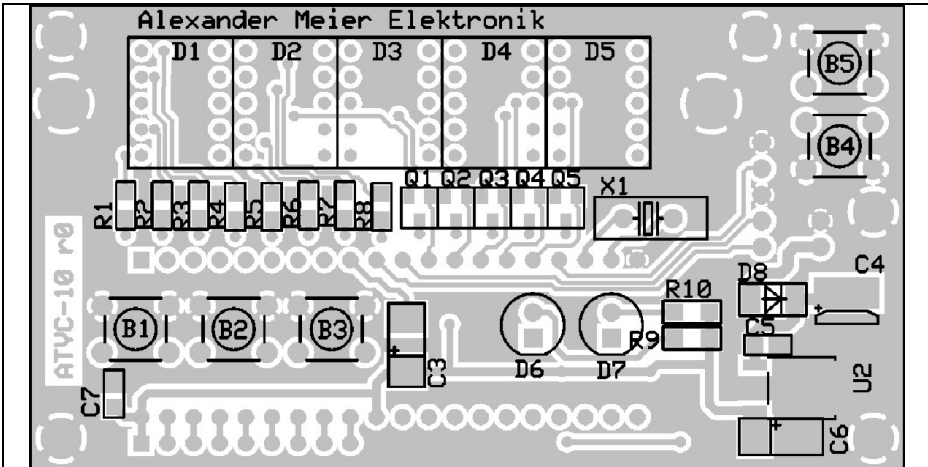


Fig 6: Top side component layout for ATV transmitter input module.

pressed, the band can be traversed quickly. As soon as the key is released, the ATV transmitter resets its frequency. The frequency display flashes until the ATV transmit module has confirmed the new frequency.

Preferred frequencies can be stored using the three station keys, M1 to M3. For this purpose, the frequency to be stored is first selected, using the Up/Down keys, and then the stations key is held down for

more than 1 second. As soon as the frequency has been stored, the display flashes briefly. To call up a stored frequency, just briefly press the corresponding stations key.

If the PLL is not locked (e.g. briefly, during a frequency change) it is signalled by LED D6. If an externally monitored output stage fails it is signalled by LED D7.

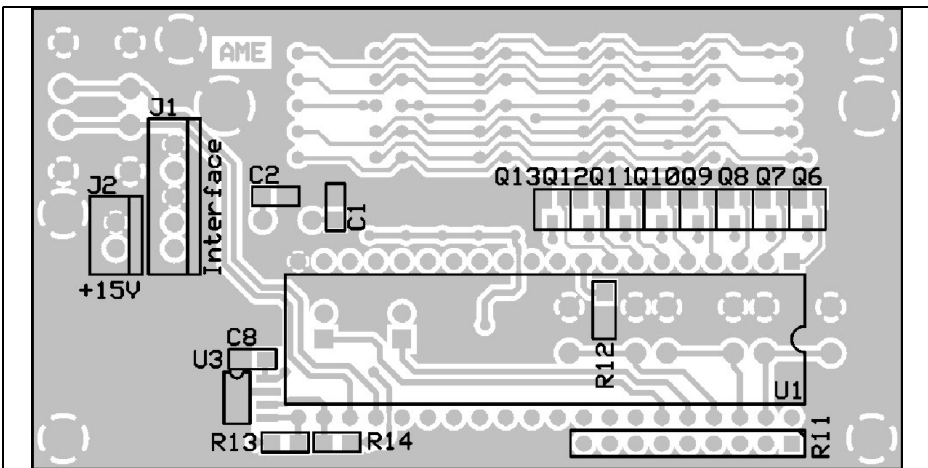


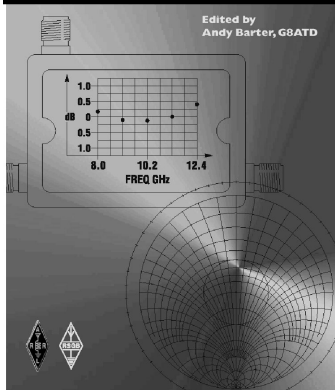
Fig 7: Bottom side component layout for ATV transmitter input module.



The International Microwave Handbook

INTERNATIONAL MICROWAVE HANDBOOK

Edited by
Andy Barter, GBATD



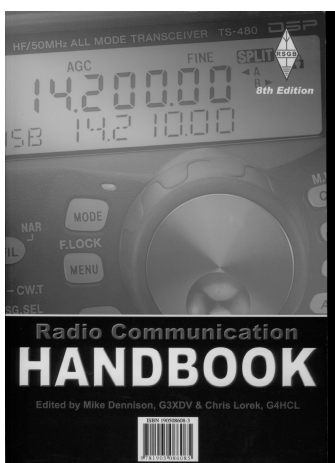
The microwave bands are an excellent area for radio amateurs who want to experiment and construct their own equipment. The RSGB in partnership with the ARRL has produced this invaluable source of reference information for those interested in this area, along with excellent designs from around the world to fire the imagination. Material has been drawn from many sources including the RSGB journal RadCom and the ARRL publications QST & QEX. Alongside this material a truly international range of sources have been used including items from Germany, Denmark, New Zealand, Slovenian and many more.

The earlier chapters of the book provide invaluable reference material required by all interested in this exciting area of experimentation. Techniques and devices are covered in depth, leading the reader to understand better the wide range of equipment and techniques now available to the microwave experimenter. This book contains a wide selection of designs using the latest technology that can reasonably be used by radio amateurs and ranges from ones that can be reproduced by most radio amateurs to those that require a high degree of skill to make.

With the explosion in consumer electronics using microwave frequencies the opportunity to experiment has never been greater and this book is simply the best guide to the area of microwave radio.

Available in The UK for £24.99 from www.rsgb.org/shop and in The USA for \$39.95 from www.arrl.org ISBN 1-872309-83-6

RSGB Radio Communications Handbook



The 8th edition of the Radio Communication Handbook has been extensively revised with several chapters completely re-written. This is the largest revision for many editions, and includes a great deal of new material. Once again, the RSGB has recruited experts on a wide variety of subjects to produce the most comprehensive guide to the practical side of amateur radio. Covering the entire spectrum from the basics through to advanced projects, and including many classic circuits, the Radio Communication Handbook makes an essential shack accessory.

The beginner will find much of value as the first few chapters explain the basics, leading to the chapters on specialist topics. For the experienced radio amateur there is a host of new ideas, including modern techniques such as microprocessors, surface mount components and computer aids to designing circuits and antennas.

* New larger format, * Nearly 800 A4 pages, * Half a million words!, * Over 1500 illustrations, * Extensively revised, * 26 chapters and two appendices, * Much new material, * Complete

PIC-A-STAR transceiver project, * Projects for 136kHz to 76GHz.

Available in The UK for £29.99 from www.rsgb.org/shop ISBN 190508608-3



Gunthard Kraus, DG8GB

An interesting program: Circuit simulation using PSPICE

1. What's it all about?

Anyone who has anything to do with electronics or circuit development has undoubtedly already heard something of PSPICE (SPICE for the PC); many people know it and some work with it every day. So we shall be dealing with the following questions here:

- What's behind PSPICE?
- What are its benefits?
- How much do you need to spend on it and how does a newcomer or a beginner get over the initial hurdles as quickly as possible ?

2. What's behind it?

To put it clearly and simply, PSPICE is a program for simulating the behaviour of electronic circuits or components in the time domain. In simple language, this means that a voltage with an easily selected waveform is applied to the input and, once a certain time has elapsed, all points on the circuit can be interrogated

to check for voltages and currents. The term "circuit" can be understood more and more broadly every day, because new models are always appearing on the Internet for everything imaginable such as DC motors or all kinds of sensors. The principles of this software were developed over 30 years ago by Berkeley University in the USA, they still monitor the standards and licenses.

We have already mentioned the keyword here; the central objective is always to obtain a model of the component to be investigated. It must either be available in the program used or have been published by the component manufacturer. A file of this nature contains all the data required for the simulation, arranged in accordance with precisely defined rules of the game (PSPICE syntax). These model files have the extension "*.cir" for individual components, whereas for extensive model collections (such as all NPN transistors from a company) "*.lib" is used.

For a rapid and almost ideal Schottky diode, for example, we have:

```
.MODEL  
.SCHOTTKY  
D(IS=1.0E-10 EG=0.69 N=1.0  
XTI=2.0 CJO=0.5PF)
```

You can already guess what this signifies

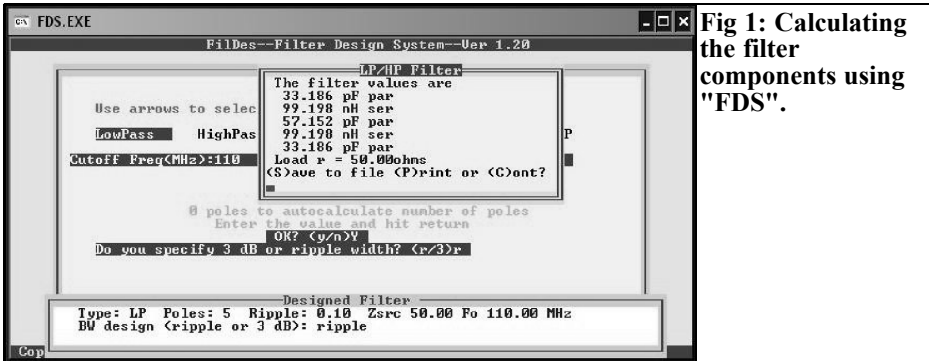


Fig 1: Calculating the filter components using "FDS".

(please take a close look: there's always a dot at the beginning of each item):

.model - gives the basic information for a new model description in a component library.

.Schottky - is the designation of this model.

D(.....) - gives the actual physical description for the simulation. The letter "D" is the identifier for a diode. In the brackets we find those details which only the specialist understands: "IS" denotes the saturation current and "CJ0=0.5 pF" reflects the depletion layer capacitance, which is not difficult to guess. For the rest though, you certainly have to pore over some clever books.

Unfortunately, this simple descriptive

structure very quickly becomes insufficient - just think of an operational amplifier, which contains many transistors and/or FETS and diodes. Here the "sub-circuit" appears as a new collective term, which is easy to identify in the model description as ".SUBCKT" and is used for many new models.

There we also find the connecting pin numbers of this component, which are needed for creating the circuit diagram.

It is extremely important that the sub-circuit description, which is often very extensive, is correctly terminated with "ENDS".

For the familiar universal diode, 1N4007, the description is shown in Table 1 (the asterisk introduces a note which is ignored for the simulation!).

Table 1: Circuit description for a universal diode type 1N4007.

*1Nxxxx diode models and subcircuits
 *rename of Thomatronik model D1N4007

```
.SUBCKT 1N4007 1 2
D1 1 2 D1N4007
```

```
.MODEL D1N4007 D IS = 1.09774E-008 N = 1.78309 RS = 0.0414388
+ EG = 1.11 XTI = 3
+ CJO = 2.8173E-011 VJ = 0.50772 M = 0.318974 FC = 0.5
+ TT = 9.85376E-006 BV = 1100 IBV = 0.1 AF = 1 KF = 0
.ENDS
```



Please examine this item carefully:

- The precise attribute description is introduced by “.MODEL” and bears the name “D1N4007”.
- This description, “D1N4007”, is used for a diode model, D1, which has two connecting pins “1” and “2”, and is generated by the line “D1 1 2 D1N4007”.
- And it's only with “.SUBCKT IN4007 1 2” that we obtain a complete component for use in the simulation program (it makes no difference at all to PSPICE what this subcircuit itself looks like - it is considered simply as a building block!). The last line “.ENDS” represents the successful conclusion of the subcircuit.

Since the components are now available, we go on to set up the circuit. Previously, this required a considerable amount of effort, you needed to use a text editor and specify line for line between which nodes (connection points for two or more components) a specific component was mounted, and what model description applied to it. This listing is known as the “Netlist” and even today every PSPICE version still needs this netlist as the pre-condition for the simulation. Only nowadays no programmers would dare ask their customers to key in all the data. People now work exclusively with convenient circuit editors and so set up their circuits on the screen. The software finally generates the necessary netlist from the circuit!

But what happens when everything has been drawn and the simulation is started? What we have is this:

First the program carries out a DC analysis and reacts promptly if it finds no DC link to earth at any of the nodes in the circuit. The user is requested to eliminate this gross error immediately via a screen

message. In practice you may have to insert a resistance of between 10 and 100 Ohms between the connection point and the earth in a series circuit of two capacitors. Messages concerning faulty connections between two components are announced in exactly the same way, if the pins are merely lying loosely next to one another. Inductances are also decidedly treacherous, since according to theory if a DC current is applied in an ideal inductance the current can sometimes be infinitely big. So never insert a coil or a transformer without also providing a series resistance (however tiny it is).

Once this hurdle has been surmounted and the program is satisfied with the circuit created, the simulation begins. The trick used for the simulation is so simple that you could almost have thought of it yourself. You take only a small step forward in time - so small that you can consider all changes occurring as linear and you can therefore get by with very simple mathematical formulae. Once everything has been calculated for this time segment, you take another small step forward, calculate the linear changes again and carry on according to this principle right to the end of the time specified for the simulation.

Of course, there is one thing you must bear in mind:

If you select a very small step in time, the voltage or current curves look perfect; but it takes up calculation time and memory space to store all the intermediate and final results. On the other hand, you can start with time steps that are so big that sinusoid signals are made up from a lot of short, straight sections and look unattractive. Here the users themselves must decide what is more important to them.

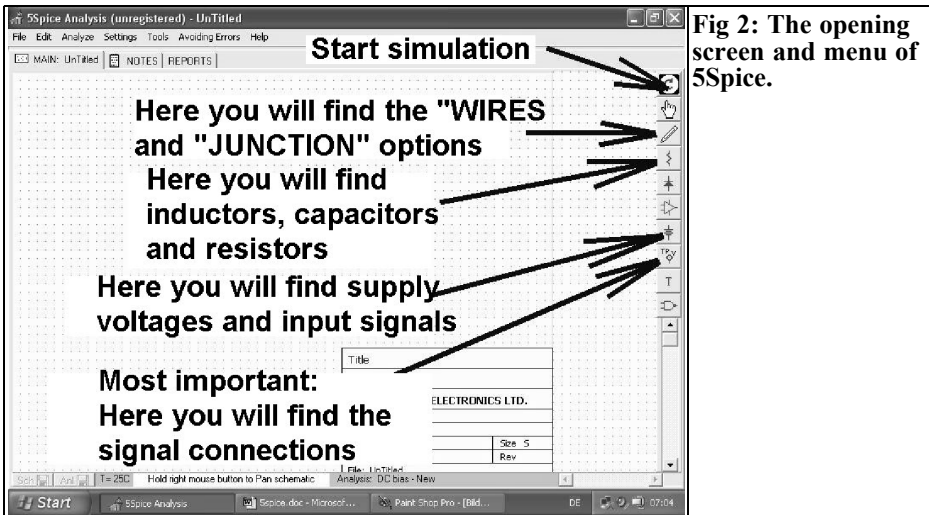


Fig 2: The opening screen and menu of 5Spice.

3. What are the benefits of PSPICE?

That's an easy question to answer: The PSPICE simulation is so accurate that the user can incorporate all the effects involved in the circuit in the form of components, and in exactly the way that each model describes the actual behaviour of each component!

But one of the biggest advantages is that PSPICE always takes account of the chronological behaviour of the signals and so automatically informs the user of any non-linearity or distortion within a circuit. The harmonics are determined by conversion into the frequency domain using the mathematical process known as the Fourier transformation. Frequency responses (referred to by the technical name of AC- Sweep) present no problem and are carried out correctly by even the simplest version of the program.

So as users we ask ourselves the question why we should actually continue to use linear S-parameter CAD programs such as PUFF if PSPICE is such an all-

singing, all-dancing solution. The answer is very simple and demonstrates that in practice either method, or even a combination of both, or even something completely new such as "Harmonic Balance" can give the desired result. The objective is actually the most accurate possible correlation between the simulation and the assembled circuit using various inputs. With PSPICE the models and the precise data that can be obtained are always the stumbling block because we have to drag around lots of data. If you are thinking about microstrips and all that goes with them, they can be more easily handled in the frequency domain and with a stock of models for PSPICE they don't (so far) look as good.

S parameters are mainly based on information that has been measured and recorded in tabular form (see the S parameter touchstone files published by all manufacturers) and for this reason they and their predictions are closer to practice. The simulation results are immediately checked against the circuit already assembled using network analysers, and modern microwave CAD programs nowadays even have an interface through which all measurement results can be fed into the simulation. You can then im-

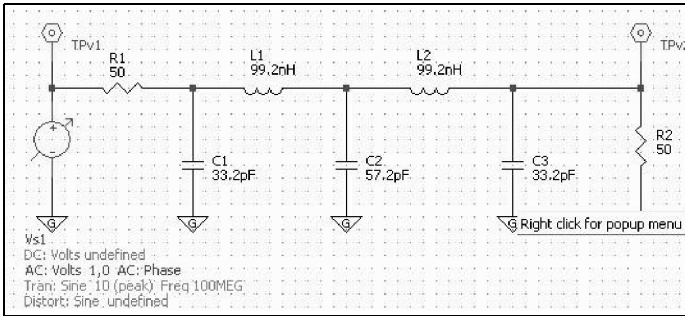


Fig 3: Circuit diagram of the Chebyshev low pass filter.

prove your results in the next round, and thus the development goals being targeted are attained very quickly and efficiently. However we can't do without PSPICE if, for example, the saturation behaviour of a transmitter output stage has to be determined at 2.3GHz, or a circuit component has to be designed, etc.

4. How can I get into PSPICE easily and effectively?

The simulation software available on the market today can be described as outstanding, after 30 years of development a very high standard has been reached, with reference both to the accuracy of simulation and to the convenience of entering and outputting data. There's just no trash left!

The reverse side of this high level is, first, the high purchase price, which can easily be as high as that of a new top of the range car. On the other hand, the modern programs are equipped with so many options and functions that either the screen very quickly appears overloaded or you have to stumble around in innumerable menus and sub-menus. All this is not exactly very encouraging for a beginner.

The software companies take a lot of trouble over their pricing policy and offer either a free 30-day trial version or a student version with restricted scope. On the other hand, people using pirated copies are often very energetically pursued (no wonder at these prices!).

The student versions can certainly justify their existence, but for serious work they seize up decidedly quickly. There is usually a limit to the maximum number of components that can be used, the amount of operating memory that can be used is restricted, or important functions and/or components are not released.

The author discovered a whole load of different solutions for those in quest of alternative simulation software:

- My teaching colleagues wanted a program that was cheap or completely free, but which nevertheless provided accurate simulations, was easy to operate after a brief familiarisation period, and could be copied without the pangs of conscience and passed on to schools or students.
- As mentioned, the component models are getting better all the time, but consequently they are also getting more expensive all the time. And that's why, even for the simulation of a simple test circuit (Op-amp as non-inverting amplifier), a student version completely failed to do the job, and merely displayed a message



“memory restricted in this version”. So for my part I also wanted the alternative to be able to process even the most complicated models at any time without protest and without the incorporation of additional models into the component library degenerating into a process which takes a whole evening.

- I also had a request from a fellow radio ham to write another article on the subject.

So I started an Internet search - but not just confined to the software companies involved in this kind of thing. I also considered people anywhere in the world who I knew via E-mail and who shared my hobby or with whom I had discussed problems. The response was truly amazing, since it was obvious that the questions I raised had also been occupying the minds of other people.

Following someone's recommendation, I came across:

4.1. The “5Spice” software

It became crystal clear that this program, which several people recommended to me, had obviously been written by someone who had had a lot to do with circuit simulation and was irritated by the complexity involved both in familiarising oneself with the big package programs and in operating them. So he took an original Berkeley SPICE Machine and fitted it into a Windows environment that was very easy to operate. Two versions can be downloaded from the Internet (<http://www.5spice.com>):

- The free “basic model” can be used immediately after installation. What's very original here is how compact it has been made, as the maximum character area for the input circuit diagram is limited to half an A4 page.
- If you pay about 150 US\$ for a licence, you not only get a maximum

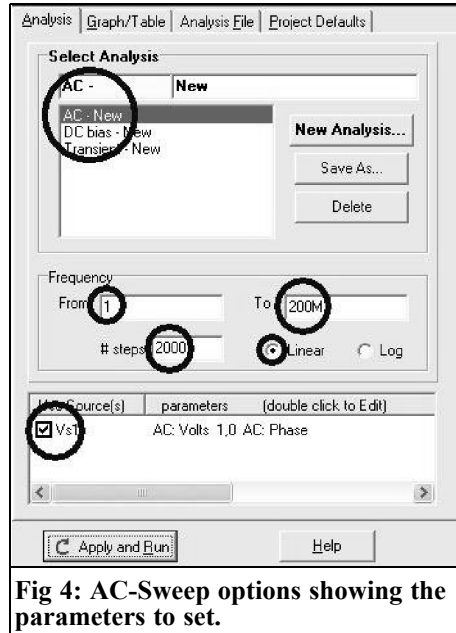


Fig 4: AC-Sweep options showing the parameters to set.

character area as big as a sheet of A2, but you also have available noise simulation, distortion analysis, Monte Carlo component scatter analysis, more digital modules and some other things. In addition, you now have support via E-mail.

It is well-known that you learn about new things best by using them yourself. So, as you see below, I tested and demonstrated the efficiency and user-friendliness on a few selected examples.

Anyone wanting to do more intensive work with the program should download a tutorial from the author's homepage which is over 50 pages long and is entitled “Fun with PSPICE circuit simulation” and which was written for any colleagues, schoolchildren, students and radio hams who are interested. The range of the circuit examples simulated in there goes from RC low pass filters all the way to the most varied rectifier circuits, and from two-stage transistor amplifiers all the way up to standard DC/ DC converters.



4.2. Let's start with something familiar: a 110MHz low pass filter

4.2.1. Setting up circuit diagram

Nothing beats practical experience, so now we just need to switch on the PC.

VHF Communications readers should find Fig 1 familiar, it is the well-known Chebyshev low pass filter with the following parameters:

- Cut off frequency = 110MHz
- Ripple = 0.1dB
- Degree of filtration $n = 5$
- $Z = 50$ Ohms

and with the following component values:

- $C1 = C3 = 33.2\text{pF}$
- $C2 = 57.2\text{pF}$
- $L1 = L2 = 99.2\text{nH}$

This has already been used a few times and in this form it supplies the free filter program “fds.zip” with all the information we shall need later.

Once “5Spice” has been successfully installed and started, we are greeted by a screen as in Fig 2. What is important is

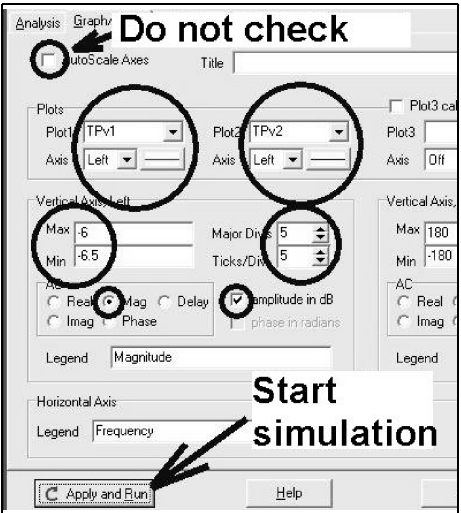


Fig 5: Parameters to set for graphical output.

the menu strip on the right-hand side of the screen. The means of accessing all components is available, together with earth symbols, supply points and dimensional markers, so we can go straight to the drawing the circuit diagram (Fig 3).

Using the circuit editor, things really go very well, and we merely have to know that the options for neat drawing (rotate,

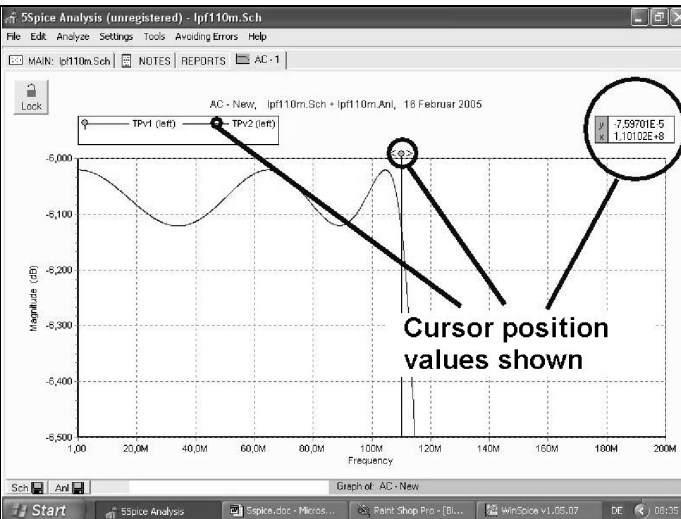


Fig 6: Result of AC-Sweep.

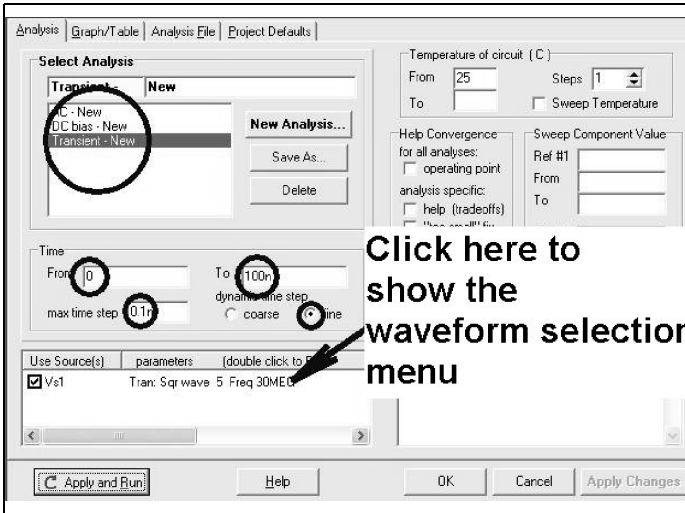


Fig 7: Parameters to set for transient analysis.

reflect, move designation, etc.) are available in the form of a selection menu if you right-click on a component. You also need to right-click when you've successfully fetched and deposited a component to free the component from the cursor.

And here's another tip:

You can't generate electrically conductive links in the circuit diagram except by using "WIRES"! True, there is also "Lines" for normal drawing, but these lines are only for decoration. If two lines intersect and you want to link them, you need the function: "JUNCTION".

If you have also moved all attribute fields of the supply point out of the way and consecutively numbered every component using a double click on its graphical symbol, the screen should look like Fig 3. Please compare everything carefully with your own design once more before going any further!

4.2.2. Determining S21 using the AC sweep

Those who take a closer look at the test circuit in question will undoubtedly get there themselves after a bit of consideration:

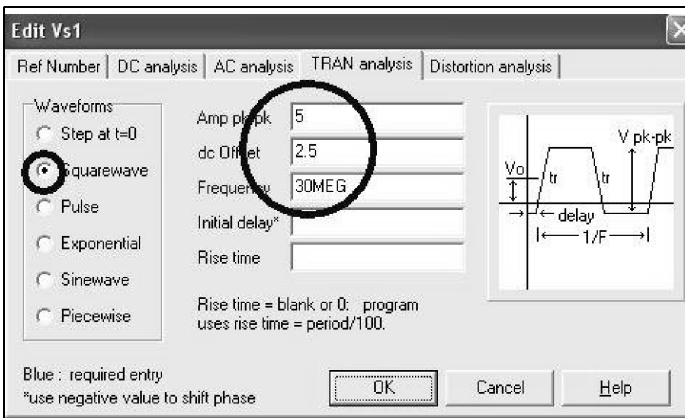


Fig 8: Parameters to set for the input waveform.

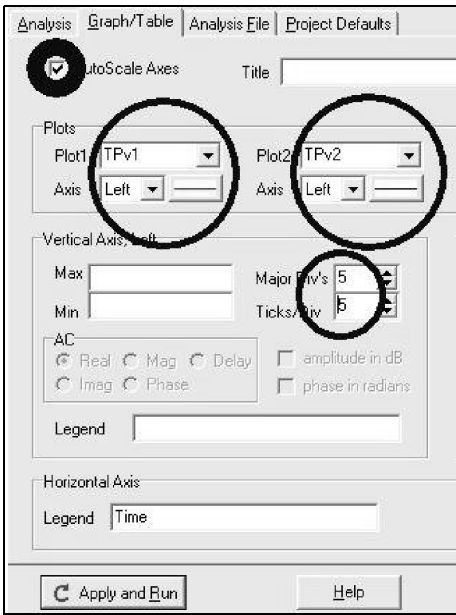


Fig 9: Parameters to set for graphical output.

First the ratio of output voltage to original voltage is simulated at various frequencies. The S parameter S21 is defined as the ratio of the output voltage to HALF the original voltage. The result is consequently too small by a factor of 2 (corresponding to 6dB). We therefore need to add on this missing 6dB in our

results (6.02dB, to be precise) to obtain the correct value for S21!

Right - now we can begin. (Fig 4):

Press the F8 key and select the Analysis tab. Select “AC New” and enter the frequency range from 1Hz to 200MHz. (A tip: the starting value shouldn't be zero, or the program will protest).

We can work with a linear frequency axis and, for example, 2000 steps (simulation points). You shouldn't forget to activate the supply point in use with a tick - right at the bottom on the left.

Then you go on to the next tab, “Graph / Table”, to programme the results output. The important settings are shown in Fig 5:

- “Autoscale Axis” must NOT be active, as we're interested in only a small range.
- “TPv1” (i.e. the original voltage) is applied to plot 1. The value should be displayed on the left-hand side. The curve itself will be represented as a continuous red line.
- “TPv2” (i.e. the output voltage) is applied to plot 2. The values are likewise superimposed on the left-

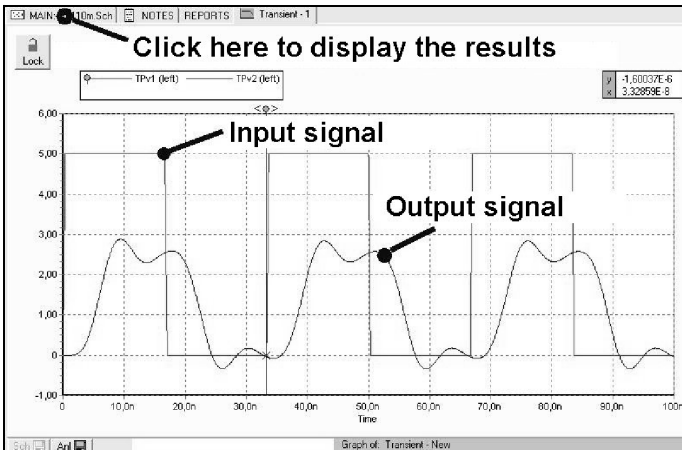


Fig 10: Results of the transient analysis.

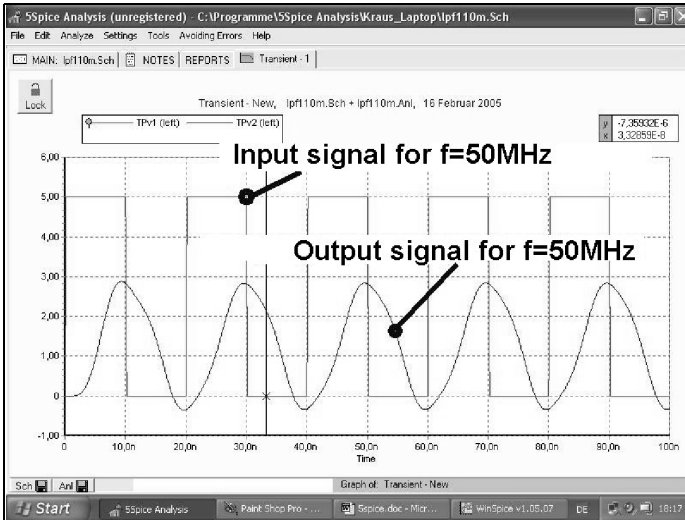


Fig 11: Results of the transient analysis with the input frequency increased to 50MHz.

hand side. The curve is shown as a continuous blue line.

- The left-hand vertical axis should show the range from -6 to -6.5dB. This range should be represented as a grid with 5 boxes. Each box is again sub-divided into five compartments.
- The “Magnitude in dB” should be displayed.

If all the settings are shown correctly, press “Apply and Run”. When the result pops up, we can lay a cursor over the

Chebyshev ripples of the filter and pull it to and fro using the mouse. The point on the curve currently active is always superimposed, with the associated frequency and amplitude values (Fig 6).

4.2.3. Transient analyses with various input signals

We could have used PUFF to get a result as good as Fig 6, or even better. It becomes more interesting if we simulate events in the time domain, since in the end that's what PSPICE is meant for.

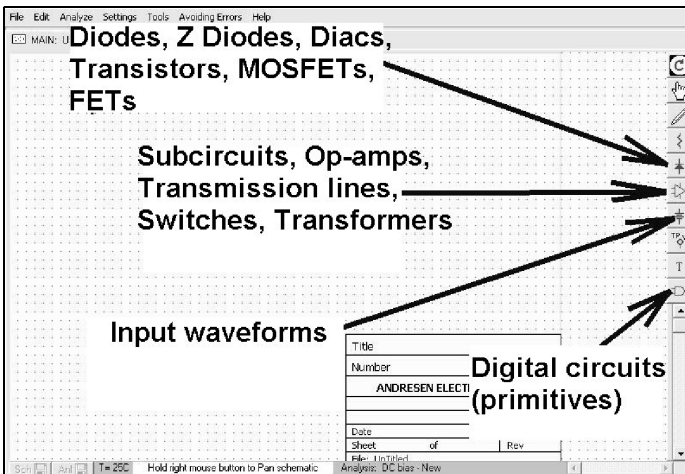


Fig 12: The range of components available for use in circuits to be analysed.

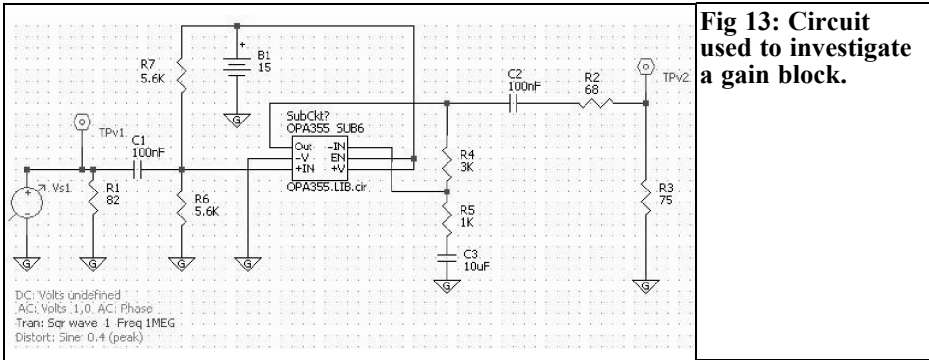


Fig 13: Circuit used to investigate a gain block.

For example, we feed in a TTL signal ($U_{min} = 0\text{ V}$ / $U_{max} = +5\text{V}$) with the frequency $f = 30\text{MHz}$ at the input of the filter to be investigated, and we convert the simulation:

Step 1

Use key F8 to select the “Analysis” tab (Fig 7). Then activate “Transient – New”, and select a simulation range from 0 to 100ns and “Dynamic Time Steps = fine” with a “max time step = 0.1ns”.

Step 2

In the bottom left-hand corner, click on the entry of the supply point and you'll get the menu as per Fig 8. There you'll see a whole series of pre-programmed curve forms. Select “Squarewave” to enter the attribute “Amp pk-pk = 5V / dc Offset = 2.5 V / Frequency = 30 MEG”.

Step 3

After confirming with "OK", we need the “Graph/Tables” tab again (if necessary, press F8 again first). What's very important now is to activate “Autoscale Axes”. The settings for the plots, TPv1 and TPv2, and for the grid divisions can be taken over from the previous example (Fig 9).

Step 4

Now click on “Apply and Run” and then Fig 10 should appear onto the screen. PUFF just can't compete any longer!

Anyone wanting to know what things look like with an input frequency of 50MHz should change to the circuit

diagram and just double-click on the supply point symbol. This also gives you the supply point attribute menu, and if you change the frequency to 50MHz you have a new simulation (Fig 11); it takes just a few seconds!

4.3. Example with integration of a new model

Those looking at Fig 12 should find themselves on the right track in the search for the components for their new circuits. A very large number of models are supplied, but with the rapid progress of technical development you can find even more if you look for models on the Internet. So we'll use the procedure in an example of this type.

4.3.1. Investigation of a gain block with the OPA355 operational amplifier

If you just want to get to know the basic behaviour of an op-amp circuit, then a

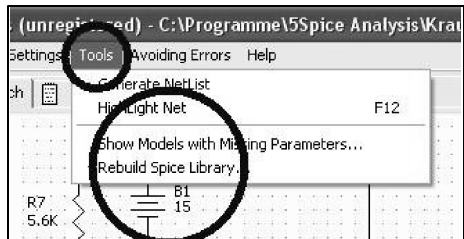


Fig 14: Option to use to incorporate the new component into the library.

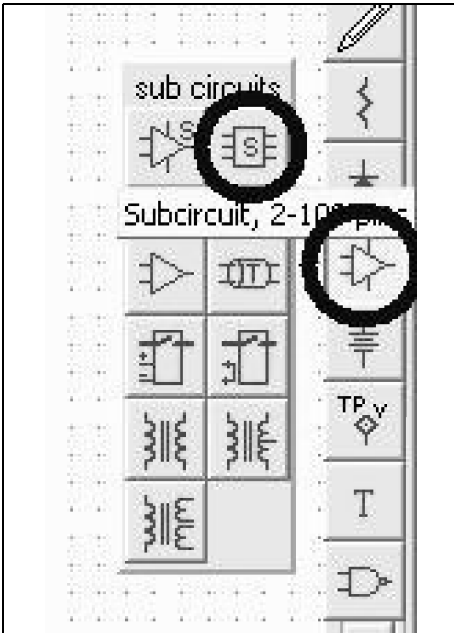


Fig 15: Components available after pressing the OP-button.

even need to bother about a supply voltage, and you can concentrate all your attention on the reactions of the circuit. Do please try it out yourself!

On the other hand, as soon as developments for a specific application are involved, things look completely different. We suddenly become interested in the behaviour of input signals with various waveforms or frequency. We can investigate the frequency response, the oscillation tendency, the influences of the supply voltage, the transient effects of spikes, together with the non-linear distortion factor at the output, the noise, etc. - and all for various types of op-amps. So repeating the simulation with another op-amp should be simple.

For this purpose, the program uses “Sub-circuits”, which describe the component used precisely and which represent a very complex circuit with many individual models (think of an op-amp with its many integrated transistors, diodes and FETS!). These subcircuit descriptions have been prepared for most modern components, in the form of SPICE models from their manufacturers.

simulation using the “Linear op-amp model” which is already in the menu will be enough to begin with. There you don't

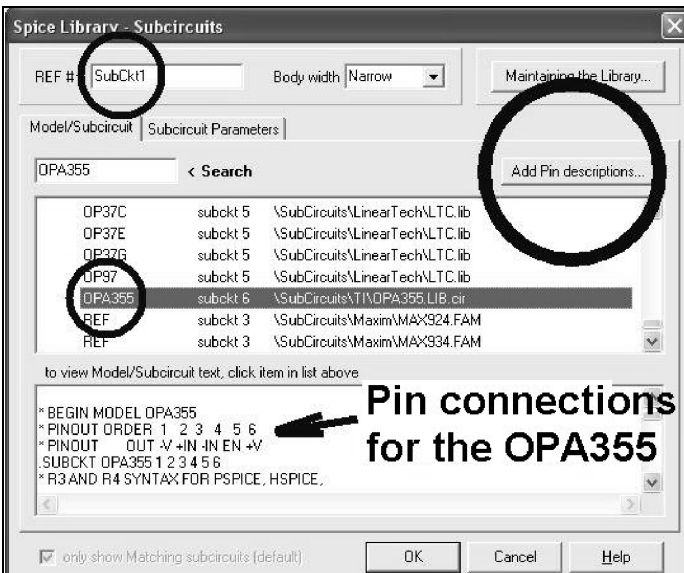


Fig 16: Selecting the OPA355.



```

* BEGIN MODEL OPA355
* PINOUT ORDER 1 2 3 4 5 6
* PINOUT OUT -V +IN -IN EN +V
.SUBCKT OPA355 1 2 3 4 5 6
* R3 AND R4 SYNTAX FOR PSPICE, HSPICE,
                    
```

Fig 17: Listing the pin functions for the Op-amp.

Let's take a closer look at the circuit as per Fig 13. Here we're dealing with an op-amp in a non-inverting circuit, which displays the following characteristics and data:

- A supply voltage of +15 V
- At the non-inverting input (“+IN”) we have half the supply voltage, generated by a voltage divider. We thus obtain a DC output voltage with the same amplitude
- We use the two resistors, R4 = 3kΩ

and R5 = 1kΩ to obtain an amplification of four

- The input resistance of the circuit is set at approximately 75Ω by means of R1
- We use the resistance R2 = 68Ω to obtain an internal resistance of approximately 75Ω.
- The entire system consequently represents a gain block, with R_i = R_a = 75Ω, which has an S21 of 6dB.
- The circuit should be usable at up to

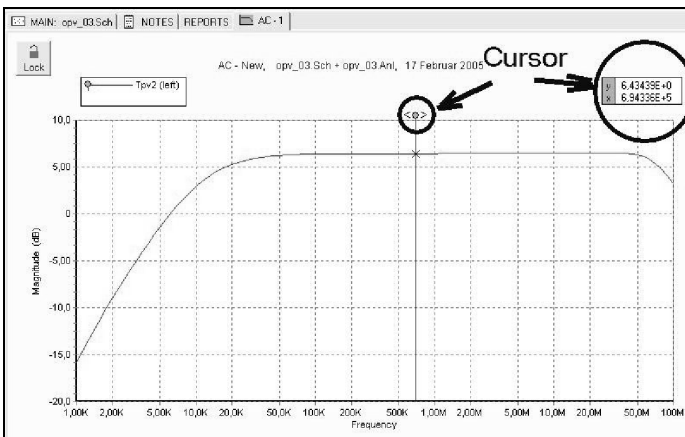


Fig 18: Result of the AC-Sweep analysis of the op-amp circuit.

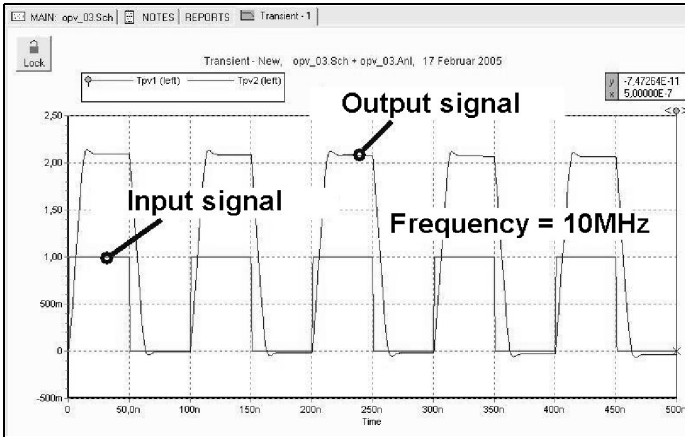


Fig 19: Performance of the op-amp with a square wave input of 1V at 10MHz.

at least 30MHz without any loss of amplification.

- The OPA355 used has an “ENABLE” connection. If this is linked to the supply voltage, the amplifier is enabled. If, on the other hand, we earth “EN”, the circuit is disabled. This makes it very easy to create an analogue switch, or even a single-ended mixer!

4.3.2. A new component for the library

By surfing the Internet for a short time using a search engine, you can obtain the SPICE model for the OPA355 and copy this file into the library of “5Spice”.

There are two directories: one is intended for “Diodes / Transistors / FETS” (basic models), while the other contains only “Subcircuits”. That’s the one the new OPA355 file has to be put into. Then start the program, go to the “Tools” menu, and click on “Rebuild Spice Library” (Fig 14). That’s really all, since the rest follows, once the component has been positioned, and it goes like this:

Step 1

Click on the “OPV Button” on the right-hand side of the image. This opens the path to “2-pole to 100-pole subcircuit” (Fig 15). Click on this and the component will be displayed on the screen. Right-click and the positioning procedure is

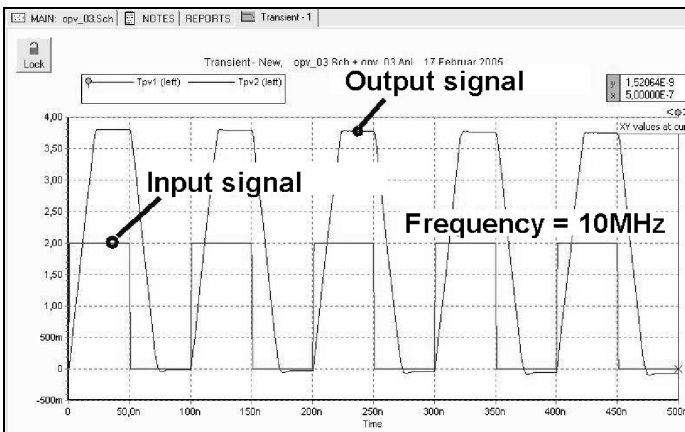
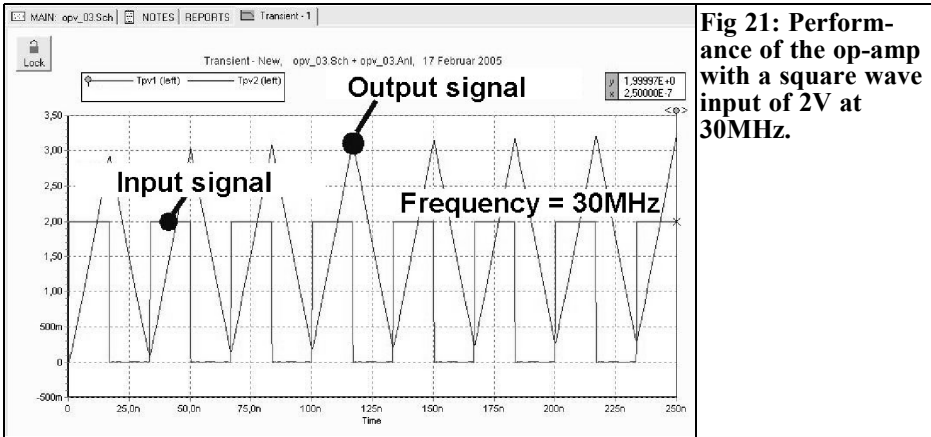


Fig 20: Performance of the op-amp with a square wave input of 2V at 10MHz.



completed.

Step 2

Double-click on the graphical symbol for contact units and switching devices, which opens the attribute menu of the subcircuit (Fig 16). Initially, this component is given the name “SubCkt1”, and then you search the stock list for the OPA355. Click on its name in the list, which loads the associated SPICE model file, and we can immediately see something very important in it; the layout of the connection pins!

Step 3

Click on “Add Pin descriptions”. Now you just need to transfer the individual pin functions, one after another, into the list that pops up (Fig 17). That’s it!

4.3.3. Investigating the circuit

Now we have to set up the simulation circuit diagram in accordance with the template from Fig 13 and wire it up. Then we call up the analysis menu with F8, and initially we just programme an AC sweep for the frequency range between 1kHz and 100MHz (see Section 4.2.2.). The result can be seen in Fig 18 and one is inclined to say; “not at all bad!”.

What’s interesting now is the reaction to a square-wave input signal where $U_{min} =$

0 V , $U_{max} = 1\text{ V}$, frequency = 10MHz (procedure: see Section 4.2.3.).

The behaviour of the gain block can be seen in Fig 19; thoroughly satisfactory. Only when we double the input voltage amplitude (Fig 20) and then treble the frequency (Fig 21) does the circuit noticeably “lose its fizz”.

At which point we can only recommend that you do a bit more research work on your own!

5. Summary

PSPICE is now an indispensable tool for a professional circuit developer at work, but unfortunately the costs for the software are at levels which only development departments can afford. So this article is intended to encourage even those with a low budget for their hobby to make it a rule to try their own projects out using this modern aid before even reaching for the soldering iron.

That’s why the author has prepared a zipped “5Spice CD”, which can be



downloaded from his homepage (www.elektronikschule.de/~krausg) without charge. It contains

- Most recent status of 5Spice software
- The author's own 50 page beginners' tutorial (already mentioned), with an extended introduction and many sample simulations. (note this manual is only in German at the moment - ed.)
- An original "Berkeley Spice Reference Manual" from the Internet
- A collection of "FAQs" from the 5Spice homepage
- Private collection of SPICE models collected in recent years (be careful:

when unzipped, this comes to well over 120 megabytes of data....)

The various items listed can also be examined individually or transferred to your own computer. So that should have invalidated all the excuses you may have had for not using PSPICE previously.

Good luck!

Amateur Television Quarterly

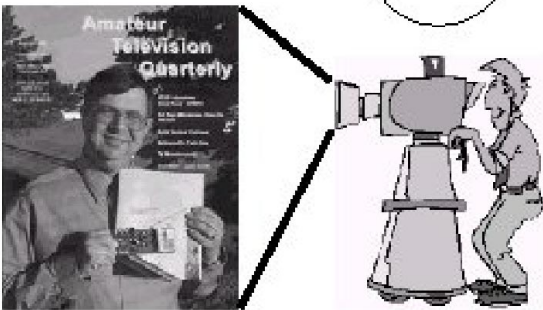
Great articles on :

ATV
BALLOONING
ATV PROJECTS
Antenna Design for ATV
SSTV
ATV ACTIVITIES
Digital ATV
ATV On The internet
WorldWide ATV Coverage

USA \$20.00 year
Canada \$22.00 year
DX \$29.00 year (US \$)

Don't miss another issue.
Subscribe Today!

VISA
M/C
AMEX



Write or check webpage TODAY for more information!
 Published by Harlan Technologies
 5931 Alma Dr., Rockford, Illinois 61108 USA
<http://www.hampubs.com>



André Jamet, F9HX

Three probes for RF measurements without DUT disturbance

1.

Introduction

RF measurements on Devices Under Test are often wrong because the measurement apparatus disturbs them. It is very common to use a probe connected to an oscilloscope. Unfortunately, even if its frequency range can be up to 100 or 400MHz, the input impedance, mainly due to its capacitance, cannot be considered too low to be negligible [1].

For example, here are some characteristics of passives probes from well-known manufacturers [2]:

- Sch : 1/10 10M Ω 12.5pF 200MHz
1/100 9.1M Ω 4pF 400MHz
- Tek : 1/10 10M Ω 12.5pF 100MHz

Active probes including a FET preamplifier offer better performances for a much higher cost:

- Ag : 1M Ω 0. pF 500MHz
- Rh : 9M Ω 3pF 3GHz
- Tek : 1M Ω 1pF 1.5GHz
100k Ω 0.4pF 4GHz

Moreover, for correct utilisation of a probe, we need to respect its load imped-

ance, i.e. the input impedance of the measurement apparatus it is connected to. Oscilloscope probes are generally designed to be loaded by 1M Ω . To connect them to a spectrum analyser or any 50 Ω input apparatus, will causes a gross mismatch. Similarly, a conventional oscilloscope cannot use a 50 Ω probe without an auxiliary load.

Professional probes are intended for qualitative (wave form) and quantitative (magnitude) signal measurements. If we are less ambitious, only to answer our main needs, we are able to make our own probes.

For example, to adjust a multi-stage frequency multiplier, we only need to maximise the desired harmonic, stage by stage, without knowledge of the actual magnitude. On the other hand, we do not want to disturb the stage's behaviour during the adjustment. Here are three probes complying with our needs.

2.

Inductive probe

EMC (Electro-Magnetic Compatibility) measurements [3] use the so-called Moebius probe. It has a small loop to pick up a magnetic field. This kind of probe can

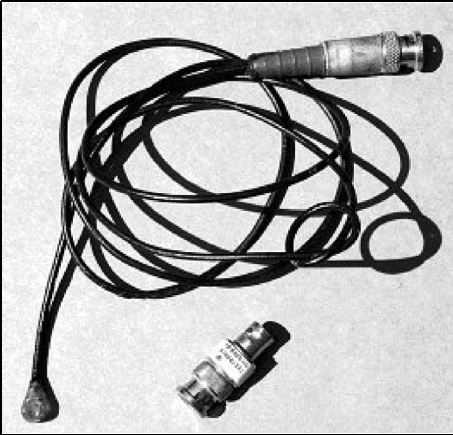


Fig 1: Photograph of the inductive probe.

shows the position of a field. Therefore, we are able to measure its frequency and trim adjustable components (trimmers, coils core etc.). We do not need to know the actual magnitude, only the variations during trimming.

The realisation is very simple: 50cm of a small size coaxial cable like RG174U terminated by a BNC plug and an about a 1cm diameter loop at the other end. That loop is made by first removing the braid on three centimetres. Slip the core by removing its insulation for a few millimetres. Solder it to the braid to make a loop. To avoid any short-circuit while using the probe, coating with two-part glue will insulate the loop and the braid. It is very simple as the photography in Fig 1 shows.

It is convenient to “sniff” a magnetic field under the case of a Neosid filter and to be able to trim each core to get the maximum output and/or the required

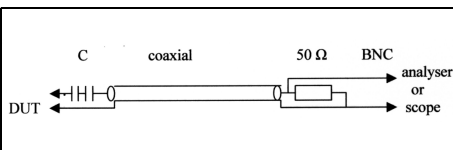


Fig 2: Schematic diagram of the capacitive probe.

bandwidth without any disturbance to the filter. To get good results and avoid any parasitic field, use a 50Ω resistor to load the inductive probe. If the measuring apparatus has a 50Ω input, a spectrum analyser for example, the connection can be direct. For an oscilloscope, we need to use a 50Ω termination between the probe and the oscilloscope input. We can use a T connector and a 50Ω load to get the job done.

Here is a practical example of the use of such an inductive probe. The device under test comprises a 106.5MHz crystal oscillator followed by several multipliers to get 10,224MHz. With a 100MHz bandwidth oscilloscope, we can see the 106.5MHz sine wave. With a 1GHz oscilloscope, the several harmonics up to 1278MHz are visible. To go further, an 18GHz spectrum analyser will show the 2556 and 10,224MHz signals.

3. Capacitive probe

With an inductive probe, it is sometimes difficult to find the right place to measure the field. A capacitive probe connected at

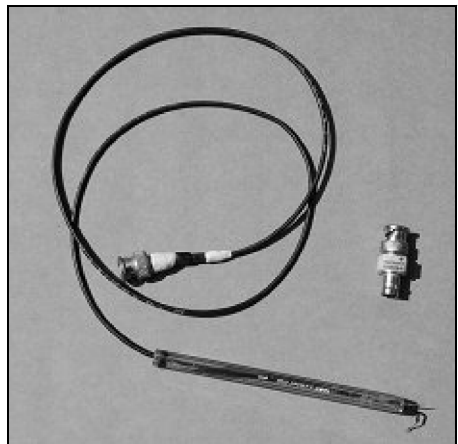


Fig 3: Photograph of the capacitive probe.

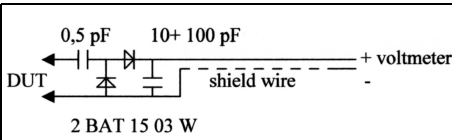


Fig 4: Schematic diagram of diode probe.

the right place can give the solution. As the current probes are not suitable, we have to make a very low capacitance probe (Fig 2). Take a piece of coaxial as before connected to a BNC at one end. We strip the braid for a few millimetres at the other end. A 0.5pF capacitor (or two 1pF in series) is soldered to the coaxial core. A 8mm pin is soldered to the other end of the capacitor. A small wire connects to the braid to get the ground connection. The whole thing fits into a ball pen body. Make sure that only the pin sticks out. Two-part glue is dropped into in the body to fix the components (Fig 3).

We now have a kind of test lead to pick up any signal. We can connect the short ground lead to the device under test's ground, by soldering or with a clip. A 50 Ω resistor must load the probe as for the inductive probe.

The frequency behaviour of a probe is difficult to calculate because of differences in manufacture. We do not claim to make precise voltage measurement; however, we can calibrate it with a signal generator. The main goal is that the measured circuit is only upsets by a 0.5pF capacitor in series with 50 Ω , which is negligible up to VHF frequencies. Above this some disturbance can occur depending of the circuit impedance. The same tests used for the inductive probe up to 10GHz are successful.

If we only want to see the signal variations and not its actual value, the ground connection is not required; this is very convenient to get a maximum output.

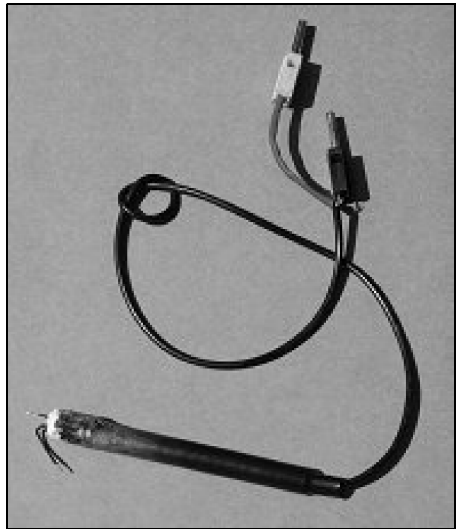


Fig 5: Photograph of the diode probe.

4.

Diodes probe

As already described, we can make a probe including two diodes as shown in Fig 4. The diode characteristics establish the frequency limit of the probe. SMD diodes and capacitors are soldered to a small PCB or are made by point-to-point wiring. All fit into a metallic envelope, made of 0.2mm copper sheet. A plastic cover is used to extend the envelope for insulation and a better handling (Fig 5).

We can use the probe without a ground connection because hand capacitance is enough. A DC voltmeter shows the output. It is better to use an analogue meter or a digital one with a bar graph to see the signal variations.

With this probe we can read a DC voltage that is not necessarily the desired signal. The reading is due to the fundamental plus harmonics. We can get a maximum, which is not the fundamental.



5.

Conclusion

It is very easy to make these probes. They are invaluable tools for us for testing and adjustment of our devices or repairs.

[2] Manufacturers Data

[3] Sniffer probe locates sources of EMI, Bruce Carsten, EDN June 4, 1998

[4] Les sondes :techniques, mesures, applications, J.C. Baud, Electronique Applications, N°4

For any more contact André Jamet, F9HX, email: agit@waandoo.fr

6.

References

[1] The Effect of Probe Input Capacitance on Measurement Accuracy, http://www.tek.com/Masurement/app_Notes/technical_Briefs/ProbeInput/eng/60W_8910_1.pdf



The UK Six Metre Group

www.uksmg.com

With over 700 members world-wide, the **UK Six Metre Group** is the world's largest organisation devoted to 50MHz. The ambition of the group, through the medium of its 56-page quarterly newsletter '**Six News**' and through its web site www.uksmg.com, is to provide the best information available on all aspects of the band: including DX news and reports, beacon news, propagation & technical articles, six-metre equipment reviews, DXpedition news and technical articles.

Why not join the UKSMG and give us a try? For more information contact the secretary: Dave Toombs, G8FXM, 1 Chalgrove, Halifax Way, Welwyn Garden City AL7 2QJ, UK or visit the website.



Wolfgang Schneider, DJ8ES and Frank Peter Richter, DL5HAT

Display unit for low power radio frequency wattmeter using an AD8362 from issue 3/2005

One of the requirements for a convenient output meter is that it should have a large display with good resolution and the ability to do the conversion from dBm to μ W, mW or W. A display unit for the low-power output meter (dBm) with an AD8362 detector in issue 3/2005 is described below. It has a multi-line digital display and an optional analogue meter (Fig1). Thanks to the micro-controller used, this system has additional options to convert and represent readings.

1. General

The measuring head with the AD8362 supplies an output voltage that is proportional to the power measured in dBm [1]. This can be displayed in various ways. To be able to represent this power in any way other than in dBm, e.g. in mW/W or even as voltage, the values have to be converted.

Using a micro-controller with the appropriate software makes the conversion of measurements easy. The results can be shown on a multi-line digital display and/or using an analogue meter.

The display unit described below makes it possible to display values in dBm, μ W/ mW/ W and μ V/ mV/ V using a four line LC display. As an option, an analogue meter can be connected, to display the power measured in Watts. The measurement ranges are defined with full scale at 1μ W, 10μ W, 100μ W, 1mW, up to a maximum of 1kW.

Since the measuring head covers only the low-power range (< 500 mW) with an AD8362, for higher power levels an attenuator or a directional coupler can be connected in series. The associated attenuation value is input to the micro-controller, which uses this value in its calculations.

2. Circuit description

The circuit diagram is shown in Fig 2. The input signal of the micro-controller assembly is the DC voltage, proportional to the measured power from the output of the external measuring head. The voltage is digitised, using an LTC1286 12-bit analogue/digital converter (IC2), with a resolution of approximately 1.22mV. Converted to the power to be measured, this corresponds to 0.025dBm per digi-

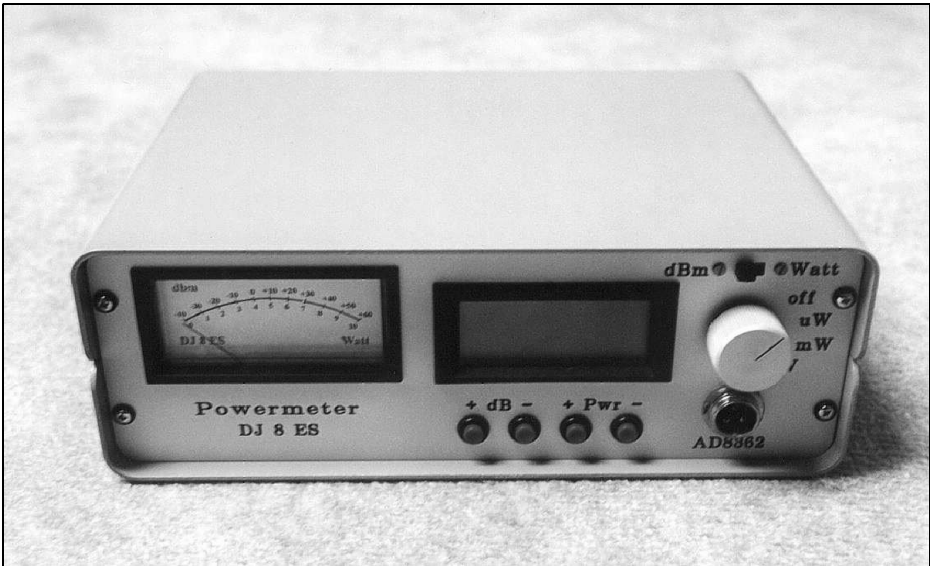


Fig 1: Photograph of the prototype display unit.

talisation step.

The reference diode LM336-5.0 gives a high precision reference voltage of 5.000V. This can be precisely adjusted by means of the trimming potentiometer R3 (10 kOhm). The reference requires its own potential free +9V power supply.

The analogue/digital converter communicates with the micro-controller (IC1, AT-Mega16) via a 3 wire bus (Chip Select, Data Out and Clock). Other ports required are wired:

- Port A: LC-Display (4-line)
- Port C: control inputs
- Port D: digital/analogue converter

The control inputs free on K4 are all LOW active, i.e. they are wired to earth.

There is an option to connect an analogue meter, e. g. a 100 μ A moving coil meter, which represents the measurement in dBm or Watts. A digital/analogue con-

verter is provided for this (IC4, LTC1257); this likewise has a resolution of 12 bits. The internal reference voltage is defined as 2.048 V.

3. Software

The software for the AT-Mega 16 bit micro-controller (IC1) was created in PASCAL using the AVRco compiler by E-LAB. This compiler is capable of multi-tasking and includes a very large number of drivers. AVR make a software with an efficient compiler available specially for ATMEL processors.

The micro-controller is working at almost full capacity with the current application software. The complex conversion of the readings into the various representations; dBm values, power in Watts or voltage in Volts, requires a high degree of accuracy and consequently a lot of

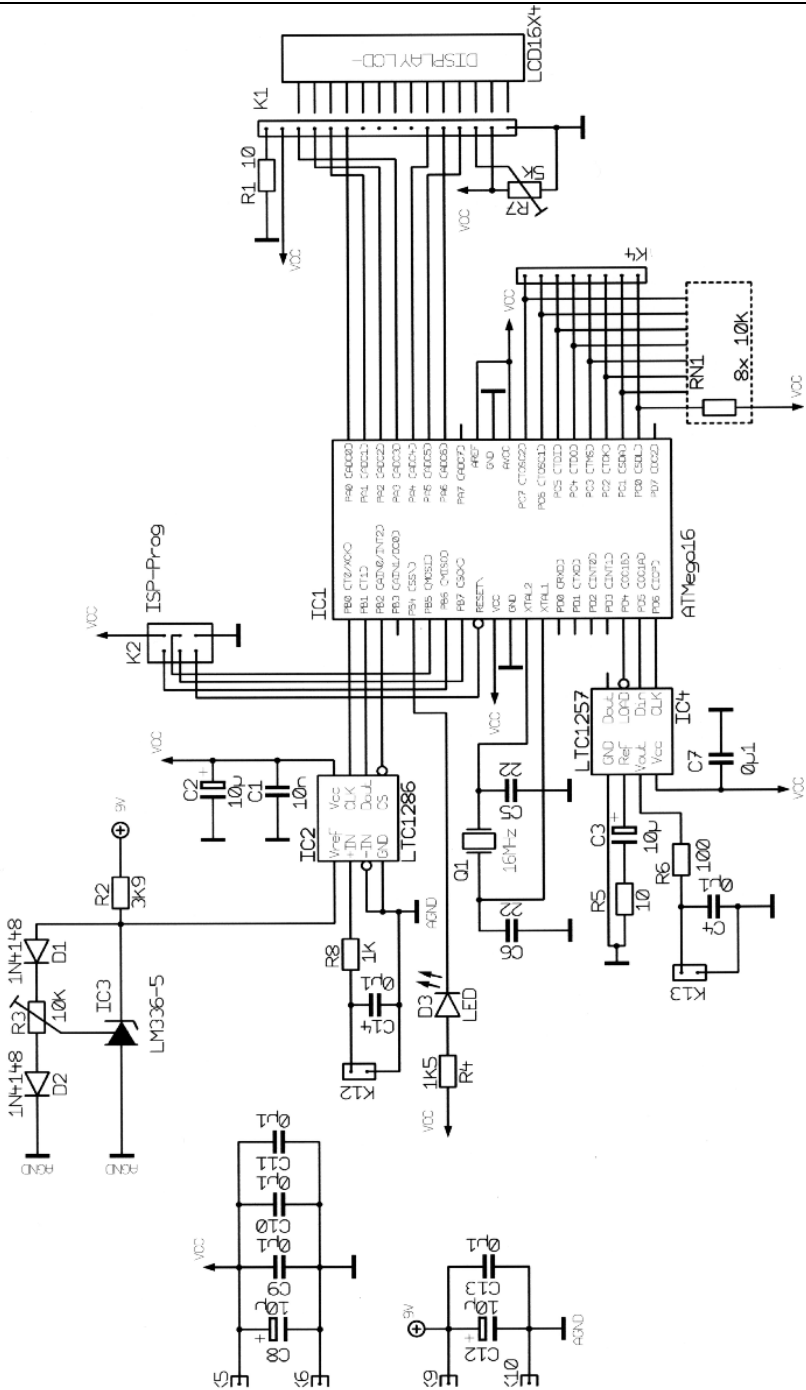


Fig 2: Circuit diagram of display unit for low power radio frequency wattmeter using an AD8362 detector.

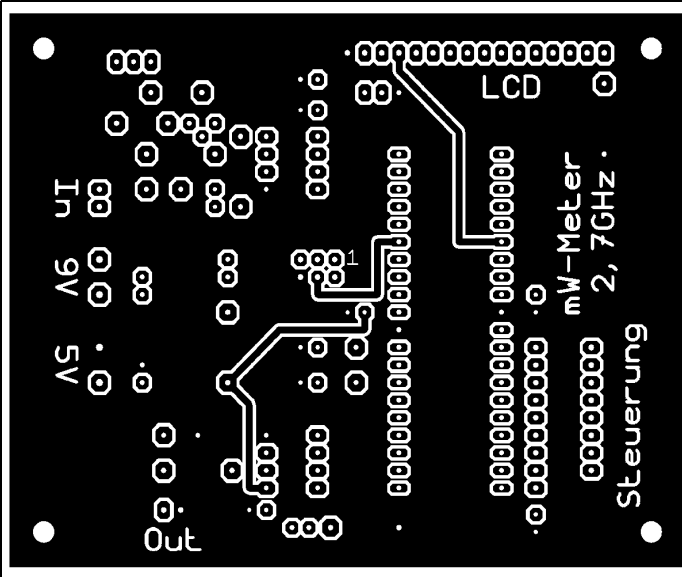


Fig 3: Top side PCB layout for the display unit.

computing effort is required. Any inaccuracies, even many places behind the decimal point, would otherwise lead to unacceptable errors in the representation of the measurements.

4. Assembly instructions

The display unit is mounted on an 80mm x 100mm double sided, copper coated,

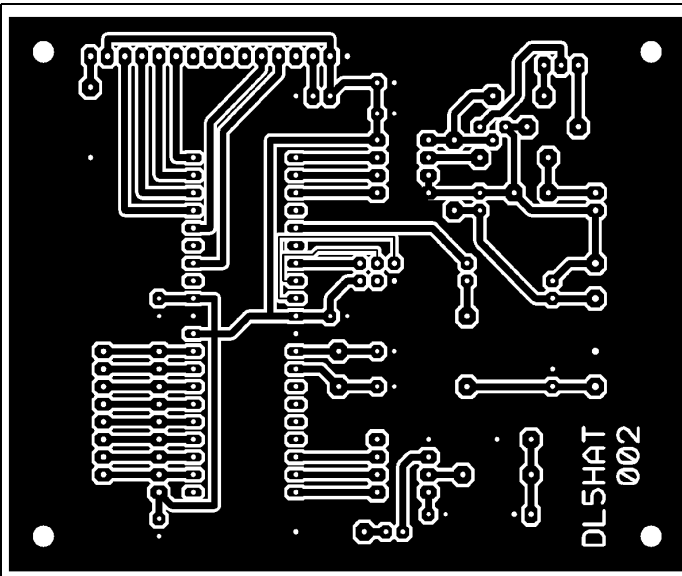


Fig 4: Bottom side PCB layout for the display unit.

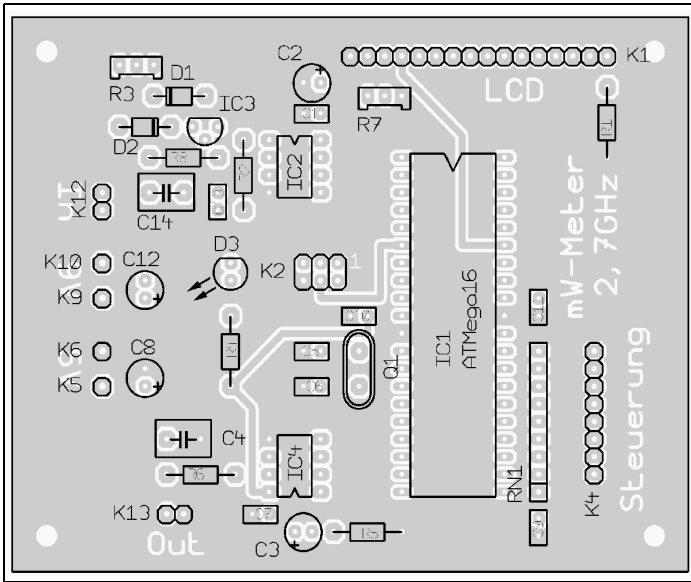


Fig 5: Component layout for the display unit.

epoxy printed circuit board (DL5HAT 002) shown in Figs 3 and 4.

The circuit board is built, in no particular order, in accordance with the components layout in Fig 5. Following a visual check for any errors, the assembly is put into operation for the first time. To do this, the supply voltage for the micro-controller (+5 V) is applied. The current consumption should be approximately 170mA. 50mA of this is used for the LC display backlight.

When the equipment is switched on, the switch-on routine should appear on the display as text. The contrast control, R7 should be adjusted. Other adjustments cannot be made until all the modules have been connected.

A power supply has been developed for the display unit the circuit diagram is shown in Fig 6 (DJ8ES 071). A 75mm x 100mm single sided epoxy printed circuit board was produced for the assembly and shown in Fig 7. The component layout is shown in Fig 8. Following a trial switching on, the output voltages of +9V, +5V

and a second +5V can be checked.

4.1. Components list for display unit

- IC1 ATmega16, Micro-controller
- IC2 LTC1286, A/D converter
- IC3 LM336-5V, Reference voltage
- IC4 LTC1257, D/A converter
- D1, D2 1N4148, Diode
- D3 LED red, low current LCD1

- LCD 16 x 4, with LED back light
- Q1 16MHz, Standard quartz
- R3 10kΩ, Precision trimming potentiometer, type 64W
- R7 5kΩ, Precision trimming potentiometer, type 64W
- RN1 8 x 10kΩ, resistance network
- C4, C14 0.1μF, Foil capacitor, RM 5 mm
- 4 x 10μF, Tantalum electrolytic capacitor, RM 2.5 mm

- K1 16-pin, plug strip
- K4 8-pin, plug strip
- K12, K13 2-pin, plug strip
- 1 x DL5HAT 002, Printed circuit board

- Resistances 1/4 Watt, RM 10mm:
- R1, R5 10Ω

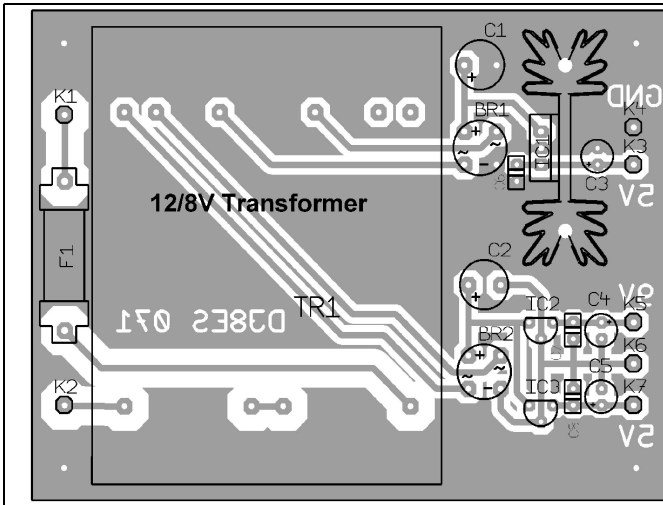


Fig 8: Component layout for the power supply.

- Schaffner
- F1 Fuse 200mA, with holder (solderable)
- 1 x Heat sink PR32/25,4
- 7 x Terminal pin 1mm
- 1 x Printed circuit board, DJ8ES 071

display ranges of μW , mW and W as well as dBm values. For this purpose, 0 corresponds to a value of -40dBm and 10 to a value of $+60\text{dBm}$. A typical scale is show in Fig 9.

5. Analogue display instrument with scale

As an option, the low-power output meter can have an analogue display meter. This is connected to the K12 output via a resistor give the correct scaling. The associated LTC 1257 DA converter (IC4) supplies a maximum output voltage of 2.048V at full scale. This corresponds to a value of $+60\text{dBm}$.

A series resistor of approximately $20\text{k}\Omega$ is required for $100\mu\text{A}$ meter. For more precise adjustment, use a combination of the fixed resistor ($18\text{k}\Omega$) and the trimmer ($5\text{k}\Omega$). For other meters the value required should be specially calculated.

A decimal scale is the obvious choice for the scale of the meter. This covers the

6. Putting into operation

Once the display unit and power supply have been completed they can be put into operation. The individual current consumption values for the operating voltages are: 5V (micro-controller assembly), 170mA, 9V (reference voltage) 1mA and 5V (measuring head) 22mA.

The reference voltage trimmer, R3, at pin 1 of IC2 (LTC1286), should be used to set the voltage to precisely 5.00V using a suitably accurate digital multimeter.

The measurement ranges for $\mu\text{W}/\text{mW}/\text{W}$ and $\mu\text{V}/\text{mV}/\text{V}$ are selected using a sequence switch. A 2 pole 4 way switch can be used for this. The same switch can be used to switch the unit on/off..

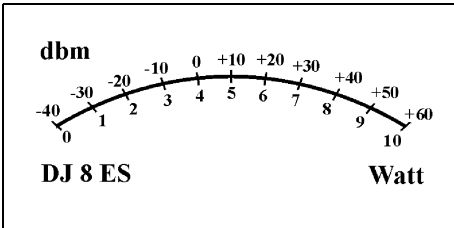


Fig 9: Sample meter scale for the display unit.

Additional control functions for setting additional attenuation (+/- dB), e.g. for using a directional coupler, and the measurement range for the analogue display (+/- Pwr) can use simple keys.

The control inputs of the micro-controller are all available on port C (PC0, PC1, ...). These are fed to the K4 socket terminal strip. Note: all connections are LOW-active, i.e. they are switched to earth. The pin configuration is as follows:

PC0:	+ dB
PC1:	- dB
PC2:	μ W
PC3:	mW
PC4:	W
PC5:	+ Measurement range
PC6:	- Measurement range
PC7:	dBm or W

7. Operational experience

The basic operation of the low-power output meter is almost self-explanatory. If a measurement range is exceeded the light-emitting diode (LED D3) is switched on. If this happens, the current display values for mW and mV are frozen, whilst the display for dBm remains unaffected.

Since the external AD8362 measuring head [1] is designed for the low-power

range (< 500mW), for higher power levels an attenuator or a directional coupler must be used. The attenuation value can be pre-set using the two +/- dB keys, the micro-controller then adjusts the displayed output. The possible adjustment range goes from 0dB to +50dB in 0.1dB steps.

The optional analogue meter normally displays measurements in dBm. This display can be switched to μ W, mW or W. The measurement ranges go up in steps of 10 (1 μ W, 10 μ W, etc.) to a maximum of 1kW. To be on the safe side, after the changeover the highest possible measurement range is selected. This can be changed using the two keys for +/- Pwr (power).

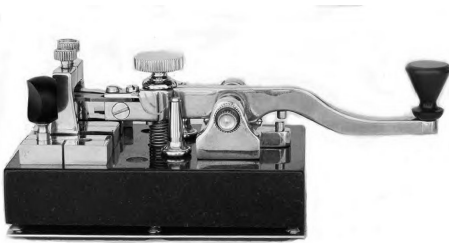
The micro-controller is programmed for use with the AD8362 measuring head described in [1]. Its characteristic output is 0.0525V per dB and delivers an output voltage of exactly 2.000V at 0dBm i.e. the output is $y = 0.0525x + 2.000$. If a different measuring head is used, Frank Peter Richter (DL5HAT) can offer individual programming for the micro-controller. Please send any enquiries by E-mail, (Frank@snokebo.de) with details of the two variables mentioned above.

8. Literature references

[1] Low power radio frequency wattmeter using an AD8362 detector, Wolfgang Schneider, DJ8ES, VHF Communications magazine, 3/2005, pp140 - 146



Introducing a new high quality morse key



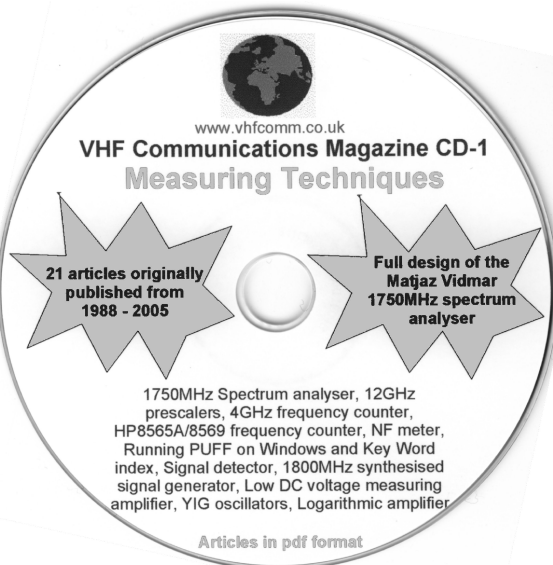
- *Real granite base*
- *Details in brass and nickel plated brass*
- *Springs and bearings stainless steel*
- *Contacts of fine silver*
- *Cables hidden with exit at rear of base*
- *Individually manufactured to order*

This key was designed and manufactured by a well known contributor to VHF Communications, Carl Lodström. It has been one of his ambitions to produce a very special morse key, this key has fulfilled this ambition. Carl showed it at Dayton 2005 and it was on the VHF Communications stand at Friedrichshafen 2005. Everyone who tried the key agreed that it is superb.

It is not a cheap key, being designed and manufactured to the highest standard, it is the Rolls Royce of Morse keys. Price £770.00

If you are interested in the key email: andy@vhfcomm.co.uk or take a look at Carl's web site: www.straightbrass.com

Compilation CD on Measuring Techniques



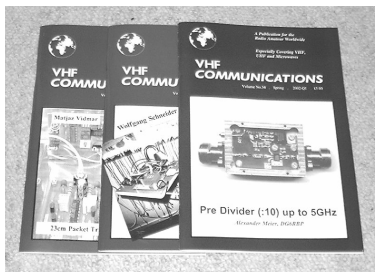
This CD contains articles on measuring techniques from VHF Communications magazine over the past 8 years. The articles are in pdf format.

The CD is £10.00 which includes 2nd class postage in The UK and surface mail overseas. Air mail postage is £0.60 for Europe and £1.00 outside Europe.

Orders can be placed via the web site: www.vhfcomm.co.uk or by fax to +44 1582 581051



Back Issues



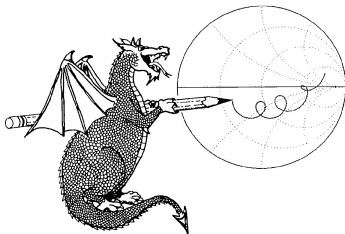
All issues ever published are now available either as photocopies or actual magazines. Issues from 1/1969 to 4/2003 are £1.00 each + postage. Issues from 1/2004 to 4/2004 are £4.70 each or £18.60 for all 4 issues + postage. See web site or page 34 of issue 1/2005 for back issue list to see which issues are only available as photocopies. There are two back issue sets that contain the available "real" magazines at a reduced price, see web site for details.

Blue Binders



These binders hold 12 issues (3 years) and keep your library of VHF Communications neat and tidy. You will be able to find the issue that you want easily. Binders are £6.50 each + postage. (UK £0.90, Surface mail £1.60, Air mail to Europe £2.00, Air mail outside Europe £3.60)

PUFF Version 2.1 Microwave CAD Software



This software is used by many authors of articles in VHF Communications. It is supplied on 3.5 inch floppy disc with a full English handbook. PUFF is £20.00 + postage. (UK £1.00, Surface mail £1.30, Air mail to Europe £1.50, Air mail outside Europe £2.50)



VHF Communications Web Site www.vhfcomm.co.uk

Visit the web site for more information on previous articles. There is a full index from 1969 to the present issue, it can be searched on line or downloaded to your own PC to search at your leisure. If you want to purchase

back issues, kits or PUFF there is a secure order form or full details of how to contact us. The web site also contains a very useful list of site links, and downloads of some previous articles and supporting information.



Gunthard Kraus, DG8GB

Internet Treasure Trove

MAXIM

A very well-known name among IC manufacturers. This company has markedly increased its activities under the heading of “Modern wireless communications technology” and massively increased the quantity of application notes available. All of this has been revised and now appears with a new look that makes it a pleasure just to go into various subjects a bit more deeply (which naturally include the “Wireless, RF and Cable” section).

Address:
<http://www.maxim-ic.com/appnotes10.cfm/>

University of San Diego / RF & Microwave Engineering

A real treasure trove for people who are interested in various RF subjects, since the pdf files available are not limited to lectures from courses. The site in fact has links to interesting articles by other people that are perfectly matched to the theme.

Address:
<http://home.sandiego.edu/~ekim/e194rfs01/>

Electronics Tutorials

Here's something from a part of the world in which people (looked at from our point of view) are standing on their heads. An Australian site, crammed full of interesting links and tutorials - so somewhere you can root and rummage around in for hours. And here's one thing you can search for straightway: just look for the Marine Corps “Field Antenna Handbook” (192 pages, downloadable as a pdf file). A handsome and interesting book!

Address:
<http://www.electronics-tutorials.com/>

Amateur radio links

There are some very industrious people in our amateur radio community, because here we have alphabetically listed links to every possible aspect of our hobby. So



this is something that you're bound to need again and again, so that you can make a search more selective and make more progress. There's a lot of hard work behind this site.

Address:

<http://www.dj6qo.de/intadar.htm>

University of Essex / Nanofab

You've guessed it already: this site deals with the cutting edge stuff in semi-conductor manufacture and nanotechnology. This site has been conceived as an information centre for people from the appropriate technological fields, but it's also very interesting for the ordinary person who's interested in electronics to have a look behind the scenes and marvel at what can be done nowadays and how it's accomplished.

Address:

<http://privatewww.essex.ac.uk/~bolat/>

Agilent

When did you last visit the homepage of the successor to HP? If it was more than 2 months ago, you should just take another look at it. The RF and microwave application notes have been greatly expanded and revised.

Address:

<http://www.agilent.com/spg/sitemap/RFmicrowave-sitemap.html>

National Instruments

By using the link below, you'll reach the "developer zone" of National Instruments, and many other paths branch out from there. Lots of literature available, but you can also access a well-constructed online lexicon of communications technology and signal processing. Just take a look around!

Address:

<http://zone.ni.com/zone/jsp/zone.jsp>

AVX

This company is mainly involved with capacitors of every kind, but it also handles inductances and couplers. Consequently, microwave technology is well represented. The site contains a huge number of documents, but there are also Spice model collections and the like.

Address:

<http://www.avxcorp.com/SpiApps/default.asp>

Datasheetarchive and Alldatasheet

Anyone looking for the data sheet for any semi-conductor component or IC should start the search with these two sites. You'd certainly have trouble finding a site where things are made so easy.

Address:

<http://www.datasheetarchive.com/datasheet/pdf/3615.html>

<http://www.alldatasheet.com/>



Spice Model Resources

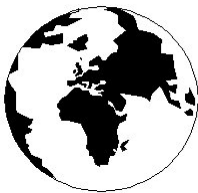
The name says it all: this is where to start your search if you need Spice models for a specific component.

Address:
<http://www.emwonder.com/spicemodels/>

Notification

Owing to the fact that Internet content changes very fast, it is not always possible to list the most recent developments. We therefore apologise for any inconvenience if Internet addresses are no longer accessible or have recently been altered by the operators in question.

We wish to point out that neither the compiler nor the publisher has any liability for the correctness of any details listed or for the contents of the sites referred to!



VHF COMMUNICATIONS

A Publication for the Radio Amateur Worldwide

Especially Covering VHF, UHF and Microwaves

Volume No.37

Winter

Edition 2005-Q4

Publishers

KM PUBLICATIONS,
63 Ringwood Road, Luton,
LU2 7BG, United Kingdom
Tel: +44 (0) 1582 581051
Fax: +44 (0) 1582 581051

Email:
andy@vhfcomm.co.uk

Editor

Andy Barter G8ATD

**VHF
COMMUNICATIONS**

The international edition of the German publication UKW-Berichte is a quarterly amateur radio magazine, especially catering for the VHF/UHF/SHF technology. It is owned and published in the United Kingdom in Spring, Summer, Autumn and Winter by KM PUBLICATIONS.

The 2005 subscription price is £20.75, or national equivalent. Individual copies are available at £5.20, or national equivalent each. Subscriptions should be addressed to the national representative shown in the next column. Orders for individual copies of the magazine, back issues, kits, binders, or any other enquiries should be addressed directly to the publishers.

NOTICE: No guarantee is given that the circuits, plans and PCB designs published are free of intellectual property rights. Commercial supply of these designs without the agreement of the Author and Publisher is not allowed. Users should also take notice of all relevant laws and regulations when designing, constructing and operating radio devices.

© KM
PUBLICATIONS

All rights reserved. Reprints, translations, or extracts only with the written approval of the publishers

Translated by: Inter-Ling Services,
62 Caldecott Street, Rugby,
CV21 3TH, UK

Printed in the United Kingdom by:
Cramphorn Colour Printers Ltd.,
15a Boughton Road Industrial
Estate, Rugby CV21 1BQ, UK.

AUSTRALIA - Mark Spooner c/o, W.I.A. SA/NT Division, GPO Box 1234, Adelaide, SA 5001, Australia Tel/Fax 08 8261 1998
BELGIUM - UKW-BERICHTE, POB 80, D-91081 BAIERSDORF, Germany. Tel: 09133-77980. Fax: 09133-779833
Postgiro Nbg. 30445-858.

DENMARK - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

FRANCE - Christiane Michel F5SM, Les Pillets, 89240 PARLY, France
Fax: (33) 03 86 44 08 82 Tel: (33) 03 86 44 06 91

FINLAND - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

GERMANY - UKW-BERICHTE, POB 80, D-91081 BAIERSDORF, Germany. Tel: 09133 7798-0. Fax: 09133 779833.
Email: info@ukwberichte.com Web: www.ukwberichte.com

GREECE - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

HOLLAND - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

ITALY - R.F. Elettronica di Rota Franco, Via Dante 5 - 20030 Senago, MI, Italy. Fax 0299 48 92 76. Tel. 02 99 48 75 15
Email: info@rfmicrowave.it Web: www.rfmicrowave.it

NEW ZEALAND - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

NORWAY - WAVELINE AB, Box 60224, S-216 09 MALMÖ, Sweden. Tel: +46 40 16 42 66. Fax: +46 40 15 05 07.
GSM: 0705 16 42 66
Email: waveline@algonet.se Web: www.algonet.se/~waveline

SOUTH AFRICA - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

SPAIN & PORTUGAL - JULIO A. PRIETO ALONSO EA4CJ, Donoso Cortes 58 5° -B, MADRID 15, Spain. Tel: 543 83 84

SWEDEN - WAVELINE AB, Box 60224, S-216 09 MALMÖ, Sweden. Tel: +46 40 16 42 66. Fax: +46 40 15 05 07
Email: waveline@algonet.se Web: www.algonet.se/~waveline

SWITZERLAND - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

UNITED KINGDOM - KM PUBLICATIONS, 63 Ringwood Road, LUTON, LU2 7BG, UK. Tel: +44 1582 581051.
Fax: +44 1582 581051. Email: andy@vhfcomm.co.uk

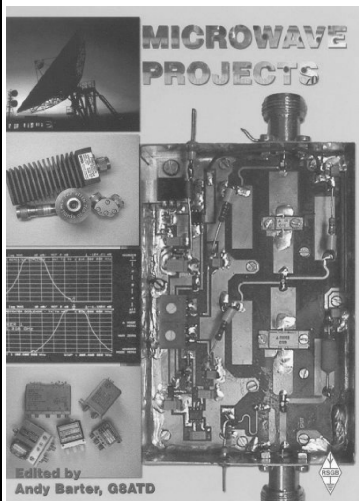
U.S.A. - GENE HARLAN, ATVQ Magazine, 5931 Alma Drive, Rockford, IL 61108, USA. Tel: +1 815 398 2683;
Fax: +1 815 398 2688 Email: atvq@aol.com

ELSEWHERE - KM PUBLICATIONS, address as for the U.K.

Web: <http://www.vhfcomm.co.uk>

ISSN 0177-7505

Microwave Projects



Microwave Projects is aimed at those who are interested in building equipment for the amateur radio microwave bands.

Packed full of ideas from around the world this book covers the subject with a variety of projects. The book has many contributors who have a wealth of experience in this area and they have produced many projects, design ideas, complete designs and modifications of commercial equipment, for the book.

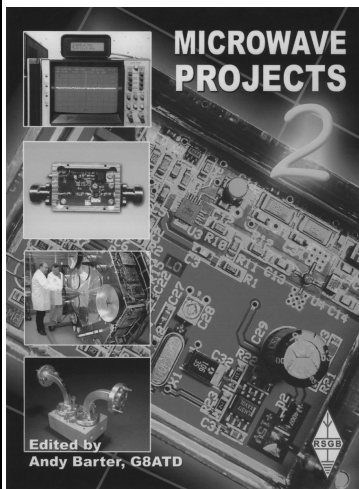
This title provides much useful information as to what can be achieved effectively and economically. Aimed at both the relative novice and the "old hand" the book also covers useful theory of designing microwave circuit and test equipment for the projects. The book includes chapters covering:

- Signal Sources • Transverters • Power Amplifiers
- Test Equipment • Design

Microwave projects is a must have book for all those who are already active on the microwave bands and those looking for interesting projects to embark on.

Available for £14.99 from www.rsgb.org/shop ISBN 1-872309-90-9

Microwave Projects 2



Following the huge success of the Microwave Projects book, Andy Barter, G8ATD brings you more innovative projects from around the world. If you are interested in building equipment for the amateur radio microwave bands, the designs in this book are sure to please you. Projects have been selected from international authors and all of the projects use modern techniques and up to date components. Details of how to obtain ready-made boards are included with most projects. The book includes chapters covering:

- Transverters and transmitters
- Receive amplifiers
- Power amplifiers
- Filters
- Miscellaneous, including rare articles on using YIG oscillators and microwave absorbers.

If you are interested the microwave bands or just in amateur radio construction Microwave Projects 2 provides great ideas and projects to satisfy everyone.

Available for £14.99 from www.rsgb.org/shop ISBN 1-905086-09-1

COMPLETE KITS, PCB's & ICs ARE AVAILABLE FOR RECENT PROJECTS

If the kit or PCB is not in this list please contact K. M. Publications

READY MADE	DESCRIPTION	ISSUE	No.	PRICE
DG6RBP	Pre Divider (:10) up to 5GHz	1/02		£ 140.00
DG6RBP	12GHz Divide by 1000 prescaler	4/03		£ 160.00
DG6RBP	12GHz Divide by 10 prescaler	1/04		£ 160.00
DG6RBP	Universal measuring amplifier for low dc voltages	2/05		£ 130.00
DG6RBP	Universal PLL oscillator	3/05		£ 118.00
DG6RBP	ATV transmitter with PLL for 10GHz	4/05		£ 193.00

KIT	DESCRIPTION	ISSUE	No.	PRICE
DJ8ES-019	Transverter 144/28MHz	4/93	06385	£ 120.00
DJ8ES-019/50	Transverter 50/28MHz	2/95	06392	£ 120.00
DJ8ES-047	Log Amplifier up to 500MHz with AD8307	2/00	06571	£ 42.00
DG6RBP-002	Pre Divider (:10) up to 5GHz	1/02		£ 115.00
DB6NT-Rotor	Simple Speed Control for Rotators	2/02	06533	£ 35.00

PCB	DESCRIPTION	ISSUE	No.	PRICE
DJ8ES-019	Transverter 144/28MHz or 50/28MHz	4/93	06384	£ 10.00
S53MV	Set of PCBs for Matjaz Vidmar Spectrum Analyser	4/98-4/99	S53MV	£ 65.00
DJ8ES-047	Log Amplifier up to 500MHz using AD8307	2/00	06569	£ 6.50
DG6RBP-002	Pre Divider (:10) up to 5GHz	1/02		£ 18.00
DJ8ES-053	Frequency Generator to 4GHz - mixer	1/02		£ 10.50
DJ8ES-054	Frequency Generator to 4GHz - oscillator	1/02		£ 9.50
DJ8ES-054	Frequency Generator to 4GHz - microcontroller	1/02		£ 15.50
DB6NT-Rotor	Simple Speed Control for Rotators	2/02	06532	£ 5.50
DB6RBP	ATV Squelch	4/02	06542	£ 3.50
DG4RBF	Set of 4 PCBs for FC4000 Frequency counter	1/00		£ 20.00
DG6RBP	Universal measuring amplifier for low dc voltages	2/05		£ 16.50

MISC.	DESCRIPTION	ISSUE	No.	PRICE
DG6RBP	Milled aluminium housing for 5GHz prescaler	1/02		£ 26.00
SP8910	SMD SP8910 divider for 5GHz prescaler	1/02		£ 40.00
DG4RBF	EPROM for FC4000 Frequency counter	1/00		£ 10.00

Please note that due to component supply problems some items have a long delivery time, please ask for advice on delivery.

Minimum shipping charge £5.00

K. M. Publications, 63 Ringwood Road, Luton, Beds, LU2 7BG, U.K.

Tel / Fax +44 1582 581051 email andy@vhfcomm.co.uk

Web Site <http://www.vhfcomm.co.uk>