VHF
COMMUNICATIONS

A New Band Pass Filter Design for µWave Projects

John Fielding, ZS5JF

Contents

Thank you to everyone who has renewed their subscription for 2008. You may have noticed that the magazine was delivered in a polythene envelope, this is because several subscribers have told me that the paper envelopes were getting damaged in the post. From the tests that I have done the new envelope will withstand the postal system much better. Because I am now using a clear envelope I have changed to a single sheet address label with the stamp printed with the label. I have also added a customer number and the expiry date of your subscription for your information.

I have had some requests for circuit diagrams and PCB layouts available on the web site. I want to provide this but only make it available to subscribers of the magazine. I have added a request form to the web site, this requires your customer number to verify that you are a current subscriber before your requests are emailed to you.

The band pass filter article by John Fielding is a mechanical construction project. Because many subscribers may not have full workshop facilities, John is going to write some short articles to help with simple workshop procedures, the first of "John's Mechanical Gems" is in this magazine.

73s - Andy

Gunthard Kraus, DG8GB

Practical project: Noise factor measurement with older spectrum analysers: part 1

The theory and practice of noise factor measurement plus HOW and WHY older measuring instruments can be used is described in detail in this article.

1.

General information

If you are developing low noise RF amplifiers, the control of noise factor is part of the project. All modern microwave CAD programs have a facility to simulate this but everyone wants to test the practical results.

There are marvellous measuring instruments but the price is a barrier for hobby use. I remember some articles in books and sample applications from the Internet that describe using a spectrum analyser for this task. The question: is an old HP140 with an HP8555 module suitable? What is missing for this measuring task, what must be developed? How do we proceed with these measurements and what theoretical knowledge is necessary? It is also necessary to know what to do, in which order and why it must be done. This article gives the answers to these questions and shows practical solutions.

2.

A short course on the phenomenon of "noise"

2.1. "Noise" - where does it come from?

That can be answered quickly and precisely: in each electrical resistance where a current flows and electrons move. As soon as heat comes into play (that is always the case above absolute zero), electrons have an independent existence. They move ever more jerkily and not straight from minus to plus. They collide, rebound, are hurled forward or off to the side…. This makes the current vary irregularly by small amounts due to the influence of heat. This effect is called Thermal Noise. Even if no outside voltage is applied these independent movements of the charge carriers, due to heat, develop a small open circuit voltage V_{noise}. It can be computed as follows:

$$V_{noise} = \sqrt{\frac{4hfBR}{e^{\frac{hf}{kT}} - 1}} \qquad (1)$$

Where:

h = Planck's constant

k = Boltzmann's constant = 1.38 x 10-23 J/Kelvin

T = absolute temperature in Kelvin

B = bandwidth in Hz

f = centre frequency of the band in Hz

R = resistance value in ohms

That seems terribly complicated and not practical, but the following can be used without problem to at least 100GHz and temperatures down to 100°K:

$$V_{noise} = \sqrt{4kTBR} \qquad (2)$$

Changing this formula around it suddenly looks much simpler:

$$\frac{\left(\dfrac{V_{noise}}{2}\right)^2}{R} = kTB \qquad (3)$$

- It is a simple indication of power! Each resistance, independent of its resistance value, produces an "Available Noise Power" proportional to kTB.

- The resistance should be regarded as voltage source consisting of V_{noise} and a noise free *internal resistance* R to generate the noise power. A noise free load resistance with the value R is connected to this source. Because voltage across the load resistance is then half V_{noise} from formula (2) the load resistance receives the Available Noise power *kTB*.

- This noise power increases linearly with the absolute temperature of the circuit and the voltage with the square root of the power. The spectral power density (power per Hz) is independent of the frequency; this is called "white noise".

Importantly:

Receiver systems are nearly always specified in terms of "power levels" instead of voltages. These are logarithmic measurements so that gain can be calculated by adding levels instead of multiplication. The most familiar unit is "dBm" that is not a voltage but a power rating in relation to the system reference resistance.

P_0 = 1 milliwatt at the system resistance

Thus:

Power level = 10log (power value / 1mW) in dBm

(4)

If the noise power "*kTB*" is considered more exactly, an interesting simplification can be introduced:

$kTB = (kT) \times B$ = (Noise power density) × (Bandwidth)

The noise power density "*kT*" represents the power in every Hz; this must be multiplied by the bandwidth in order to calculate the noise power produced. Converting this to a level calculation the following formula should be committed to memory:

Each resistance produces at ambient temperature ($T_0 = 290°K$) an available noise level and thus an available noise power density of

-174dBm per Hz of bandwidth.

For bandwidths larger than 1Hz:

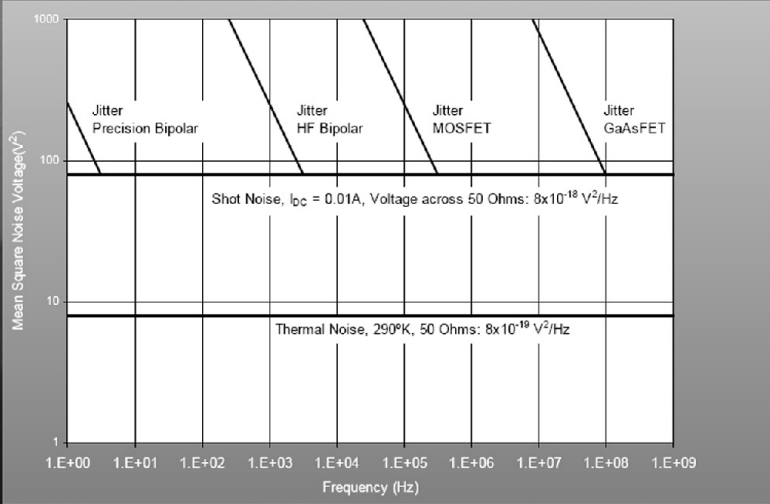Available noise level in dBm = -174dBm + 10log (bandwidth in Hz)

As a small example:

The no load noise voltage V_{noise} at the terminals of a 50Ω resistor for a bandwidth of 100kHz shall be calculated:

3

Fig 1: This diagram from [6] shows the three different noise generating mechanisms mentioned in the text.

Noise level with matching =
-174dBm + 10log (100000) =
-174dBm + 50dB = -124dBm

That results in an available noise power of :

$$P = 1mW \cdot 10^{\frac{-124}{10}} = 1mW \cdot 10^{-12.4} = 4 \cdot 10^{-16} W$$

That will produce a voltage across the 50Ω resistor of:

$$V_{noise} = \sqrt{P \cdot R} = \sqrt{50\Omega \cdot 4 \cdot 10^{-16} W} = 141nV$$

So the open circuit V_{noise} voltage is twice this value, thus 282nV.

Other references:

On the Internet there is a lot of free literature on this topic. The chapter "Noise" in the standard work on microwaves by David Pozar [1] seems to be the model reference. Unfortunately this book is very expensive and therefore a better reference is probably the most important HP Agilent application note [2]. It also contains all basic information. In addition the application notes [3] and [4] from the same company are worth downloading. The application note from MAXIM [5] on the topic of noise measurement is also worth having on your computer.

2.2. Further sources of noise

Each active component (valve, bipolar transistor, barrier layer FET, MOSFET, HEMT etc.) produces two additional types of noise:

- **Shot Noise** produced in valve diodes and pn transitions by distortions of

the current flow when crossing the potential differences. It produces wide band, white noise.

- **Flicker Noise** or "1/f – Noise" (sparkling noise) results from defects in the crystal structure caused by impurities. They lead to short pulse type fluctuations in the current flow that produce a spectrum whose power density decreases with rising frequency. There exists a corner frequency and it is interesting to see how this differs between different active components. A good impression of this can be found in Fig 1. The reference material in [6] should be obtained from the Internet, it contains a precise but compact introduction to the subject.

Before semiconductors, gas discharge tubes were used as sources of noise, they produce a very wide band plasma noise.

The following section shows how the different kinds of noise in a circuit can be considered and how they can be summarised by only one parameter.

2.3. Noise temperature and noise factor for a two-port system

When components of a communications system are specified they usually have the same system resistance (normally 50Ω, but for radio, television and video systems, 75Ω) and signal quality information required for correct transmission. This is often expressed by the SINAD value (Signal to Noise and Distortion). The maximum signal level is limited by over modulation, distortions, intermodulation etc. The noise "floor" (in addition, other disturbances e.g. cross modulation from other channels) gives the minimum signal level in the system. Therefore each component must be specified by its S parameters a well as its noise behaviour.

When regarding the noise, the following data is possible and usual (unfortunately not all authors use the same terminol-

ogy):

- The equivalent noise temperature of a component expresses the self-noise of the component by an additional temperature rise for the source resistance. The component itself is thought of as noise free.

- The Noise Figure "NF" indicates, in dB, how much worse the signal-to-noise ratio becomes after the signal has passed through the component (this is a measure of how the noise from the internal resistance is added to the signal).

- The Noise Factor "F" is like the Noise Figure as above, BUT not in dB, but as a simple power relationship.

2.4. The noise temperature

Consider a two-port component, for example an amplifier for a 50Ω system. The signal source must have an internal resistance of 50Ω in order to give a power match with the 50Ω input impedance. If the input signal has an amplitude of zero, therefore only noise is present at the output.

Now we proceed in two steps:

- First consideration: the amplifier is noise free; it sees only the internal resistance of the source. This has a temperature of T_0 = 290°K (standardised ambient temperature world-wide) therefore the noise signal supplied to the matched amplifier input is:

$$P_{N\_IN1} = kT_0B$$

- Second consideration: the source resistance is cooled down to zero degrees Kelvin and therefore produces no noise. The input signal is still zero. Everything that is measured at the amplifier output is noise that comes from the amplifier itself. Now consider that the amplifier circuit with all its components is noise free

and the measurable noise comes just from an additional heating of the source resistance. The apparent temperature necessary for this is called "equivalent noise temperature T_e" and the associated power can be calculated by:

$$P_{N\_IN2} = kT_e B$$

The amplified sum of these two noise powers can be measured at the output (this is permitted, because the two noise parts are not correlated and may be added). The formula for the power amplification of such a component is the linear relationship of the two powers, thus the "linear power Gain" is:

$$G_{P\_LINEAR} = \frac{P_{OUT}}{P_{IN}}$$

This must be calculated if only the dB power amplification is given:

$$G_{P\_LINEAR} = 10^{\frac{G_{P\_dB}}{10}} \qquad (5)$$

Thus the entire available noise power at the component output is:

$$\begin{aligned} P_{N\_OUT} &= G_{P\_LINEAR} \cdot (P_{N\_IN1} + P_{N\_IN2}) \\ &= G_{P\_LINEAR} \cdot (kT_0 B + kT_e B) \\ &= G_{P\_LINEAR} \cdot k \cdot B(T_0 + T_e) \end{aligned}$$

This shows a very beautifully thing concerning the additional temperature rise of the source resistance. If the ambient temperature T_0 is rearranged it gives:

$$P_{N\_OUT} = G_{P\_LINEAR} \cdot kT_0 B \cdot \left(1 + \frac{T_e}{T_0}\right) \quad (6)$$

If this expression is divided by the power amplification G_{P_LINEAR}, it gives the effective noise power at the amplifier input :

$$P_{N\_IN\_TOTAL} = \frac{P_{N\_OUT}}{G_{P\_LINEAR}} = kT_0 \cdot \left(1 + \frac{T_e}{T_0}\right) \quad (7)$$

It becomes clear that the 1 in the brackets represents the ambient temperature noise from the source resistance. The other term indicates how much noise comes from the amplifier and must be added (e.g.: if the equivalent noise temperature of T_e is the same as the ambient temperature T_0, the input noise performance will be doubled). Thus the bracketed term is the noise factor "F".

2.5. The noise factor F and the Noise Figure NF

The noise factor is:

$$F = \left(1 + \frac{T_e}{T_0}\right) \qquad (8)$$

If the equivalent noise temperature of a component is required and the noise factor F is known, the conversion of this formula is child's play:

$$T_e = T_0 \cdot (F - 1) \qquad (9)$$

Therefore *a noise factor* F = 1 results in an equivalent noise temperature of zero Kelvin and designates a completely noise free component (= only the amplified noise of the source resistance appears at the output).

It is very often practical to express the noise factor in dBs. This "Noise Figure NF in dBs" is determined as follows:

$$NF = 10\log(F) = 10 \cdot \log\left(1 + \frac{T_e}{T_0}\right) \quad (10)$$

Unfortunately there are considerable discrepancies for this in literature, but we want to adhere to the English habit in this article:

F = noise factor *(linear relationship)*
Noise Figure NF = 10 log (F) in dBs
(logarithmic relationship)

A practical note for a passive component:

The Noise Figure NF of a passive component corresponds its attenuation in dBs

$$F_{TOTAL} = F_1 + \frac{F_2 - 1}{G_{P\_LINEAR\_1}} + \frac{F_3 - 1}{G_{P\_LINEAR\_1} \cdot G_{P\_LINEAR\_2}} + \frac{F_4 - 1}{G_{P\_LINEAR\_1} \cdot G_{P\_LINEAR\_2} \cdot G_{P\_LINEAR\_3}} \cdots \cdots \quad (11)$$

A 20dB attenuator has a Noise Figure of 20dB this is easy to explain. Assume that an antenna is directly connected to the input of a receiver. As well as the information signal the receiver input receives the self-noise signal from the internal resistance of the antenna. If a 20dB attenuator is connected between the antenna and the receiver the two antenna signals (noise and information signal) will be weakened by 20dB. but the receiver sees the internal resistance of the 50Ω attenuator that creates noise from the ambient temperature i.e. kT_0B. The noise level at the attenuator output in relation to the divided down portion suddenly increases by 20dB and the Noise Figure increases also by 20dB according to the definition!

Here is a summary of the most important parts of this chapter:

The Noise Figure (*NF*) indicates by how many dBs the signal-to-noise is worsened when we go through the component or system (note: the source supplies not only the desired information signal, but also noise that comes from its internal resistance).

Converting the dB value of the Noise Figure into a linear relationship gives the Noise Factor (*F*).

2.6. Series connection of components

In practice it is usual to deal with a whole chain of components or building blocks where each one strengthens and weakens differently. How can the noise behaviour of the complete system be calculated? It can be done in the following way:

Each individual component of the chain has its own self-noise (usually the Noise Figure NF in dB) and its power amplification (power gain in dB) is described by the S parameter S21. A signal voltage with an information signal level S_{IN} *and* an internal resistance, whose value agrees with the system resistance, are applied to the input of the chain. Now two things are of interest:

- How large is the information signal level at the output this gives the overall gain of the arrangement?
- What are the S/N values at the input and the output? By what factor and dB value has the S/N worsened at the output as opposed to the input?
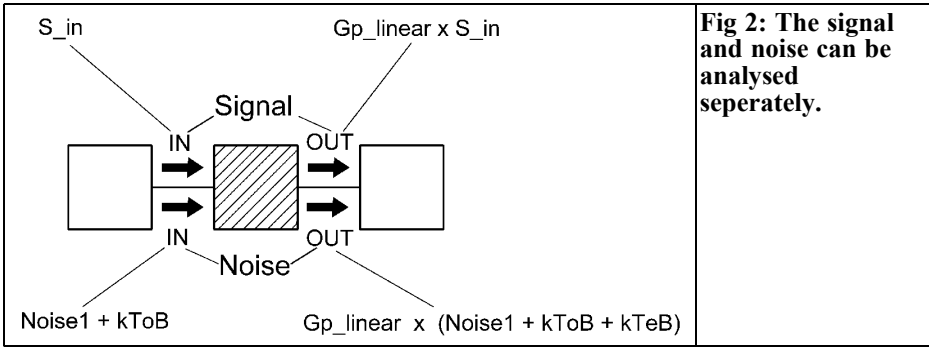
The first question can be answered quickly because it only requires the addition of the gain values (in dB) of all building blocks involved, considering the sign, in order to calculate the overall gain. The sum of input level in dBm and overall gain in dB results in the output level in dBm.

The second question is considerably more complex, because here we MUST NOT use the logarithmic values.

The correct result is only obtained using the linear values (Noise Factor F in place of Noise Figure NF in dB and power amplification G_{P_LINEAR} in place of power gain in dB). This gives: formula 11.

This shows the well-known fact that only the noise of the first stage is all included in the total noise factor. Notice the second stage contribution is less because of the higher gain of the first stage. The stages following on have insignificant noise contribution with sufficiently high gains.

Please remember: the entire noise is calculated by this formula as all being in the overheated source resistance while the entire system following is accepted as noise free.

Fig 2: The signal and noise can be analysed seperately.

In the figure, the signal path: S_in → Signal → Gp_linear x S_in; the noise path: Noise1 + kToB → Noise → Gp_linear x (Noise1 + kToB + kTeB)

2.7. Dealing with a noisy signal

This section deals with a practical question that is either treated very badly in textbooks or not at all:

How does the signal-to-noise ratio of a signal at the output of a component (with a power gain G_1 *and* a Noise Figure NF_1) worsen if the input is already a noisy signal? (This means that the noise floor is already above the theoretical minimum of kT_0B).

This situation is always true for a building block WITHIN a system chain e.g. the signal-to-noise change in an if amplifier due to a signal from the preceding mixer or, analysing the second or third stage of a 3 stage satellite LNB before the mixer.

From the previous section, the effect of the self-noise of a component is smaller if the preceding component has enough gain to overcome to noise but this must be properly proven.

Looking at Fig 2 in detail it can be seen that the component concerned receives several input signals:

- The information signal S_{IN} including the noise signal $Noise_1$, supplied by the preceding stage
- The noise power kT_0B from the source resistance (internal resistance of the preceding stage*)*

So the output signal can be divided into 4 signals:

- The information signal amplified by G_{P_LINEAR}
- Three noise powers that can be added since they are not correlated. They are: the noise from the previous stage amplified by the gain (G_{P_LINEAR} x $Noise_1$), The amplified noise from the source resistance *(G_{P_LINEAR} x kT_0B)* and the amplified self-noise referred to the input of the component (G_{P_LINEAR} x kT_eB).

The signal-to-noise ratio at the output can be written:

$$\left(\frac{S}{N}\right)_{OUT} = \frac{G_{P\_LINEAR} \cdot S_{IN}}{G_{P\_LINEAR} \cdot \left(Noise_1 + kT_0B + kT_eB\right)}$$

$$= \frac{S_{IN}}{Noise_1 + kT_0B + kT_eB} \quad (12)$$

If the expression in the denominator $Noise_1 + kT_0B$ is removed (the portion that is supplied to the component input), the answer required is achieved:

$$\left(\frac{S}{N}\right)_{OUT} = \frac{S_{IN}}{\left(Noise_1 + kT_0B\right) \cdot \left(1 + \frac{kT_eB}{Noise_1 + kT_0B}\right)}$$

$$= \frac{\left(\frac{S}{N}\right)_{IN}}{\left(1 + \frac{kT_eB}{Noise_1 + kT_0B}\right)} \quad (13)$$

Changing the formula around gives:

$$\left(1 + \frac{kT_eB}{Noise_1 + kT_0B}\right) = \frac{\left(\dfrac{S}{N}\right)_{IN}}{\left(\dfrac{S}{N}\right)_{OUT}}$$

This is a familiar expression that represents the linear form of the statement already made:

The signal-to-noise ratio (in dB) at the output of the component decreased by the Noise Figure NF (in dB).

Calling the term in the left bracketed the current noise factor F_{actual} gives:

$$F_{actual} = 1 + \frac{KT_eB}{Noise_1 + kT_0B} \quad (14)$$

This formula can be validated easily:

- The supplied noise $Noise_1$ is zero. This gives the familiar relationship for the noise factor F:

$$F_{actual} = 1 + \frac{T_e}{T_0}$$

- The supplied noise $Noise_1$ is much larger than the noise produced in the stage and the noise of the effective source resistance for this stage at the input. Then the value of the second term in formula (14) is zero therefore:

$$F_{actual} = 1$$

That is also logical, because a high noise level in front of the stage means that the small self-noise does not need to be considered.

The unfortunate fact remains that the dB values of gain cannot be used they must be converted into linear gains.

That was the theory; now the practice follows!

3.

Determining of the noise figure F

In principle this can be carried out using three different methods with appropriate variations:

First method:

Some years ago there was a noise-measuring instrument available. The noise power at the output was adjustable and the scale of the instrument was calibrated in KT_0 or dB This transmitter was connected to the input of the test specimen and a wattmeter was connected to the output. Now the transmitter was adjusted until the output noise power of the test specimen rose by 3dB (DOUBLED POWER). Therefore the input noise power was equal to the noise produced in the test specimen, this could be accurately read from the transmitter. Simply, but effectively!

The only problem if you find one of these at a flea market is that they use a special valve noise diode. These are rare parts and age very quickly. For a problem free "gem" look for similar equipment with a more modern semiconductors noise generators.

Second method (Gain method):

Measure the noise power at the output of the test specimen with a low noise spectrum analyser and the input terminated with the system impedance (usually 50Ω). If the test specimen has sufficient high gain, the self-noise of the analyser can be neglected. For the power output:

$$N_{OUT} = (Gain) \times (kT_0 Bandwidth) \times F \quad \text{in W}$$

The advantage of this method is only apparent if the power level is changed to dBm. The formula is then:

$$N_{OUT}(dBm) = (Gain) + \frac{-174 dBm}{Hz} + 10 \cdot \log(Bandwidth) + NF$$

To make the measurement the gain of the test specimen in dB must be known. The spectrum analyser displays the power output in dBm on the screen and by using the controls the power can be read. The remainder is now a simple dB calculation:

$$NF = N_{OUT} - (Gain) - 174\, dBm - 10 \log(Bandwidth)$$

Remember that this method assumes that the self-noise of the spectrum analyser can be neglected. Therefore it only works correctly for test specimens with high self-noise or very high gain

Third method (Y factor method):

The internal resistance is connected to the test speciemen input at two different temperatures, T_{hot} and T_{cold} and measure the noise power at the output in both cases.

Working with two temperatures is no joke; using liquid helium, for the low temperature and boiling water for the high temperature! Receiver measurements can be made with the antenna pointed towards the cold sky and then towards the warm earth (there have been articles in VHF Communications Magazine about this in the past). Alternative, some decades ago, special thermionic diodes or gas discharge tubes were used.

Modern noise figure measuring instruments use this principle, however special semiconductor noise diodes, usually avalanche diodes, are used. These devices produce ambient temperature noise when switched off. When they are switched on they produce a known noise power corresponding to a hot resistor. This cold to warm difference is indicated as the ENR value in dB (Excess Noise Ratio).

The measuring procedure is quite simple. With T_{HOT} and T_{COLD} the following formu-

lae are used for the output noise power:

$$N_{OUT\_COLD} = k \times Gain \times Bandwidth \times (T_{COLD} + T_{equivalent})$$

$$N_{OUT\_HOT} = k \times Gain \times Bandwidth \times (T_{HOT} + T_{equivalent})$$

These are two equations for two unknown values i.e. for the equivalents noise temperature $T_{equivalent}$ of the test specimen and for the Gain Bandwidth Product (which is of less interests). Mathematics and the microprocessor in a modern Noise Figure meter have no problems in calculating the equivalent noise temperature and then the noise figure.

$$F = \left(1 + \frac{T_{equivalent}}{T_0}\right)$$

A further calculation step gives the Noise Figure in dB.

$$NF = 10 \cdot \log(F)$$

4.

Noise measurements using your own older spectrum analyser

4.1. Measurement principle

Unfortunately the gain method cannot be used with older spectrum analysers because of their high self-noise and limited sensitivity. Therefore a different procedure must be used:

- A low-noise wideband preamplifier is used in front of the spectrum analyser. A maximum Noise Figure of 3 to 4dB as well as a gain of at least 40 ... 60dB is required. The output resistance must correspond as exactly as possible to the system resistance. S11 and S22 should not be larger than -10 to –15dB in order to exclude errors due to reflections and miss matches. Using this amplifier the

noise level arriving at the analyser input is increased so that the analyser self-noise no longer plays a role.

• The input of the preamplifier is terminated with a 50Ω resistor and the noise signal output is set to the centre of the spectrum analyser screen. Decrease the video bandwidth of the spectrum analyser until the noise floor is just a straight line this is now:

THE LINE THAT REPRESENTS THE NOISE POWER kT_0B OF THE RESISTANCE

This not only eliminates the self-noise of the additional preamplifier and the spectrum analyser but also simulates (as required) a noise free termination.

• The remainder is very simple and clarified by an example:

The 50Ω termination on the input is replaced with a test preamplifier with 10dB gain and the 50Ω termination connected to the input of this test preamplifier. The screen display will increase by exactly 10dB if the test preamplifier is noise free. If the display increases for example by 11.5dB the output S/N has decreased by 11.5dB - 10dB = 1.5dB which is a Noise Figure of 1.5dB.

The fascinating thing is that the frequency response of the additional preamplifier and the analyser play no role since all fluctuations of the gain are balanced by the calibration procedure but the minimum gain present must overcome the self-noise of the spectrum analyser...

This method is only successful if:

The gain of the test specimen (in dB) is very very accurately known. It also relies on the accuracy of the spectrum analysers logarithmic amplifier.

There are extremely high demands placed on the design of the preamplifier (absolutely oscillation free wide band with minimum 40dB but up to 60dB gain

and a good match at the input and output). It is a headache to design but with a lot of experience of designing microwave amplifiers it is possible.

To be continued

X.

Literature for part 1

[1] David Pozar, "Microwave engineering", Prentice Hall, Englewood Cliffs, N.J. 07632. ISBN 0-13-586702-9

[2] HP Agilent Application note 57, parts 1/2/3 (Download from the Web)

[3] HP Agilent Application note 150, parts 1/2/3/4 (Download from the Web)

[4] HP Agilent Application note 1354: "Practical Noise Figure Measurement and analysis for Low Noise Amplifier Design" (Download from the Web)

[5] Maxim Application note 2875: Three Methods of Noise Figure Measurement (Download from the Web)

[6] Mohr Associates, Inc.: Tutorial "Mohr on Receiver Noise". (Download from the Web)

Aristoteles Tsiamitros, DD5FT

Filter synthesis using LTspice

The modern developer has the great advantage of the circuit simulation. A simulator can be used to test new ideas plus compute and analyse components. Simulators that are based on SPICE can be used to simulate both linear and nonlinear circuits. The free full version of the SPICE simulator from Linear Technology, LTspice/SwCAD3, is briefly described in this article.

An LC filter is used in the following synthesis to describe the use of LTspice. It is possible to define your own functions in LTspice e.g. functions to compute the components for Butterworth and Tschebyscheff filters. The S parameters of the filters can be computed and shown using the network analysis in LTspice.

1.

Introduction

SPICE (Simulation Program with Integrated Circuit Emphasis) was developed in the 1960's at Berkeley University [1].

Since that time many companies have developed very good simulators based on SPICE. Apart from the full versions there are often free student versions that are

inferior, not in regard to the functional range of the full version but the number of logic elements that can be used are limited.

The SPICE simulator from Linear Technology is a full version. In addition to the usual SPICE functions an easy and fast analysis of DC-DC converters is possible with LTspice/SwCAD3.

The program (LTspice/SwCAD3, referred to as LTspice for brevity in the following article) can be downloaded from the homepage of Linear Technology [2]; or directly from [2A]. The downloaded file "swcadiii.exe" is a self-extracting file that carries out the installation. An icon called "SwCAD III" is installed on the desktop; double clicking on the icon starts the program.

Among other things installed in the software folder there is a subfolder of examples, it is worthwhile rummaging around in these examples. In the folder "\lib\sub" you will find some partial circuits. Symbols representing circuit elements are in the folder "\lib\sym".

Support for LTspice can be found in the on-line help that can be downloaded as a pdf file from [2B].
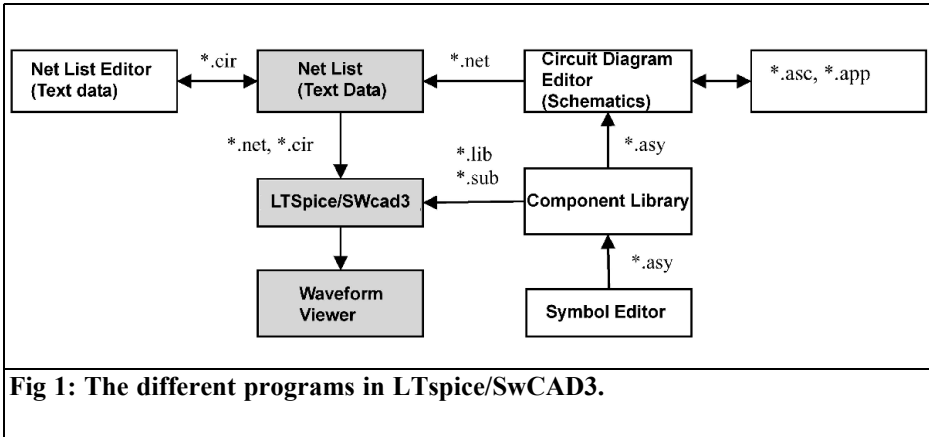
Fig 1: The different programs in LTspice/SwCAD3.

2.

The programs contained in LTspice

Fig 1 shows the logical connection between the different programs in LTspice.

Naturally the core of the programs is the simulator. The input file for the simulator is the net list. This is a text file that can be created with any text editor. LTspice has its own integrated net list editor (Fig 2).

As normal when using Windows an existing net list file can be opened using "File/Open" and typing the name of the file: *.cir, *.net, *.sp. If a new name is entered the following question is asked: "Do you want to create a new net list file?", answering "Yes" produces a new file.

As an alternative to the net list, the circuit can be entered using the circuit diagram editor integrated into LTspice (Fig 3). The circuit diagram gives good documentary value but a better insight into SPICE is gained using the net list.

An existing circuit diagram is opened with "File/Open", and typing the name of the file: *.asc, *.app ". A new circuit diagram can be created with: "File/New

Schematics". Further details about creating a circuit; where to find the components, defining and starting the simulation, are in the examples that follow.

Instructions for the simulation can be entered using the button "op" at the right hand end of the toolbar (Fig 3), these are added to the file e.g. "tran 3m" carries out an analysis of the circuit for the time interval of 0 to 3ms.

All currents and voltages that are computed during the simulation can be displayed with the Waveform Viewer. The Waveform Viewer is loaded automatically after the simulation. The dialogue "Add Traces to plot", on the menu "Plot Settings/Add trace" lists the available signals. The output voltage and the current in the coil L1 of the DC/DC converter in Fig 3 are shown in Fig 4.

LTspice has a large component library from the most diverse applications. The user can extend the library with further components from Linear Technology or from other manufacturers. Symbols for new components can be created with the integrated symbol editor. The symbol editor is loaded with "File/New symbol".

Fig 2: A net list in the net list editor of LTspice/swCAD3.

3.

Butterworth low-pass filter

In this section a Butterworth low-pass filters is simulated. The example is a 4th order low-pass filter (Fig 5), the circuit and the file are described.

3.1. Calculation formulas

The standardised elements of a filter with a given characteristic impedance can be calculated with the formula:
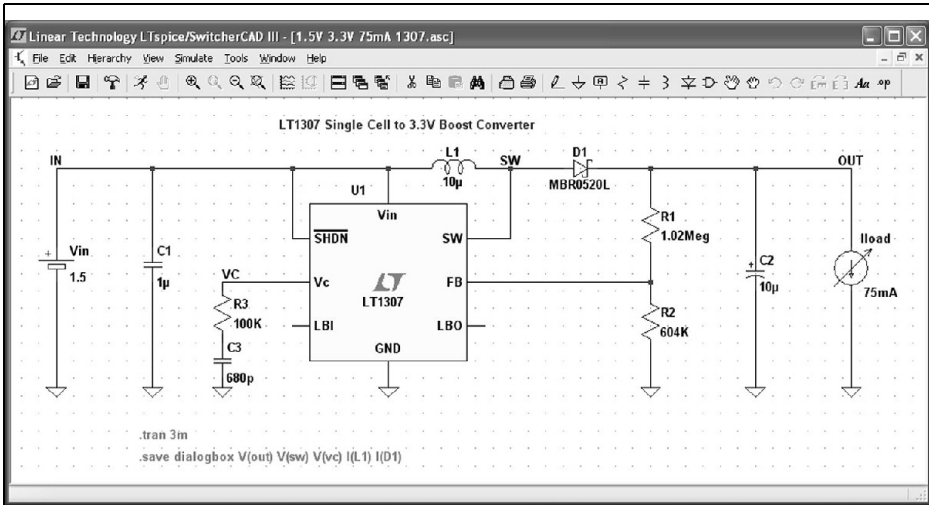
$$g_k = 2\sin\frac{(2k-1)\pi}{2n}$$

n is the order of the filter and k = 1 …. n is the run index that indicates the element.

De-standardisation is carried out using the cutoff frequency, $\omega = 2\pi f$ and the characteristic impedance w, using multiplication of standardised inductances:

$$LL(w,f) = \frac{w}{2\pi f w}$$

and the standardised capacitances:

Fig 3: The Circuit Editor (an example from Linear Technology).

$$CC(w, f) = \frac{1}{2\pi f w}$$

The generator resistance, and the terminating resistance are equal to the characteristic impedance. Rser = {w} assigns the value of the parameter w to the generator resistance. The value {w} is also assigned to the termination. Thus only the line "param" needs to be edited if a new filter is specified.

3.2. Circuit design

Start with "File/New Schematics" and an empty work sheet is displayed. The most frequently used components are shown in the symbol border. Very fast operation can be achieved using keyboard shortcuts. The key allocation (e.g. R for a resistor, F3 for wiring, F5 for deletion, etc.) can be found on keyboard shortcut map opened from the menu "Tools/Control Panel/Drafting Options/Hot key".

All components can also be found in the component list accessed by the button "Component" (or F2). For the voltage supply, select the symbol "voltage". Re-

sistors in the European format are in the list [Misc].

Clicking with the right mouse button on the voltage supply on the work sheet opens the dialogue "Independent voltage source V1". For analysis over a frequency range a sweep generator source is required, i.e. the source must be activated for AC analysis. It is sufficient to set the amplitude of the source "AC amplitude" to 1V. To set the internal resistance of the source use "Series Resistance", equal to the characteristic impedance {w} of the filter.

In LTspice the series resistance of coils is preset to approximately 1mΩ. The reasons for this are described in the online help; they concern fast algorithms for the simulation of DC/DC conveters.

For the 4th order Butterworth low-pass filter example the loss resistance of the coils would give a return loss. Odd order Tschebyscheff low-pass filters are discussed later that have finite values of ripple. Tschebyscheff low-pass filters normally have a return loss of approximately 20dB.

15
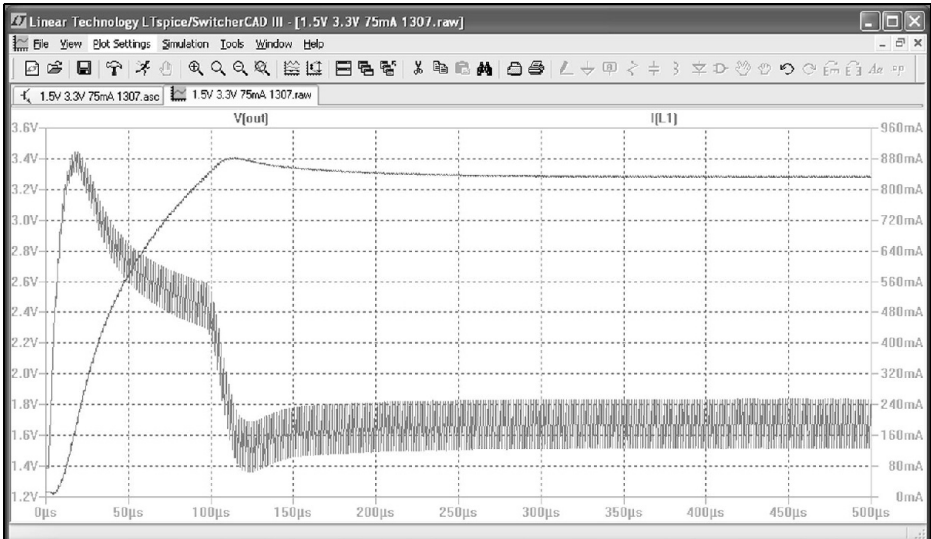
Fig 4: The Waveform Viewer showing the output voltage and the current in the coil L1 of the DC/DC converter in Fig 3.

The preset values can be used to switch off the loss resistance of the coils used. In addition a right mouse click on each coil opens the properties menu, the option "Series Resistance [Ω]" can be set to 0 for an ideal coil or the actual loss resistance.

3.3. Programming

In LTspice there is an instruction to define ".func", for definition of user functions (see online help, Dot Commands, .func, P. 67). The functions are familiar e.g. the function:

.func pythag (x, y) (sqrt (x*x + y*y))

This calculates the square root of the sum of the squares of x and y, Pythagoras. The function value can be assigned to a component e.g. a resistor, a coil or a capacitor.

.param u=3 v=4;
        are parameters that can be
        handed over

R1 N020 N021 pythag (u, v);
        Calls the function pythag ()

The resistance R1, connected between nodes N020 and N021 in the circuit

would have a value of 5Ω.

This procedure is the same as with any other programming language. First a function is defined and is then called in the program with certain parameters. In the example the call took place during the value assignment to the resistance R1.

Many mathematical functions are available e.g. sin (), cos (), sinh (), asinh (), etc., they can be used in user defined functions (see manual, Waveform Viewer, Waveform arithmetic, P. 42).

Using the SPICE instruction "param w = 50 f = 1e6", as well as ".param n = 4" sets the characteristic impedance w = 50Ω, cutoff frequency f = 1MHz and the filter order n = 4. Furthermore there are functions for the de-standardisation LL (x, y) and CC (x, y), as well as the functions for the computation of the standardised components g1 (x), g2 (x), etc.

To understand the script you must understand the difference between the declaration of the functions with the indefinite parameters x and y on one side, and the

```
        L1                      L3
      {LL(w,f)*g1(n)}         {LL(w,f)*g3(n)}

                                                    R2
                     C2          C4                 {w}

   V1                {CC(w,f)*g2(n)}   {CC(w,f)*g4(n)}
   AC 1 0
   Rser={w}
```

.ac dec 1000 1 4meg
.net I(R2) V1

1) Characteristic impedance and cutoff frequency
   .param w = 50 f = 1e6
2) Order n of the filter
   .param n = 4
3) Functions for de-standardisation
   .func LL(x,y) (x/(2*pi*y))
   .func CC(x,y) (1/(2*pi*x*y))

4) Standardised values of components
   g1: .func g1(x) (2*sin(1*pi/(2*x)))
   g2: .func g2(x) (2*sin(3*pi/(2*x)))
   g3: .func g3(x) (2*sin(5*pi/(3*x)))
   g4: .func g4(x) (2*sin(7*pi/(3*x)))

Fig 5: Circuit diagram and script for 4th order Butterworth low-pass filter.

call of the functions with the actual parameters w, f and n on the other side.

The function calls take place from the components of the filter. Thus the coil L1 calls the function LL (w, f) with the parameters w = 50 and f = 1e6, and the function g1 (n) with the parameter n = 4. The product of the two function values is assigned to the coil as inductance.

The called functions could call for further functions used for the Tschebyscheff filters described later.

The circuit diagram is shown in Fig 5. For filters of higher or lower order the circuit and the script must be edited starting from point 4). Do not forget to update the parameters n, w and f. The text editor can be opened by right clicking on the text to be edited.

3.4. Simulation

To display a frequency response with LTspice the ".AC" analysis is used. It is started from the menu "Simulate/Edit simulation Cmd". This opens the dialogue "Edit Simulation Command". The relevant parameters are shown, for the filter in Fig 5 the sweep generator was adjusted from 10kHz to 10MHz. The number of frequency steps per decade was set to 1000.

The result of the AC analysis is a frequency response diagram. For RF work the transmission factor S21 and the return loss S11 (in dB) are normally used to evaluate a circuit. The SPICE instruction ".net" calculates the network parameters, as well as the S parameters the Y and Z parameters are calculated. The instruction is entered using the ".op" button or with the key "S". In the dialogue "Edit Text on the Schematic" select the option "Spice directive", and enter the text: ".net I(R2) V1". Details of the network analysis are in the online

1) Characteristic impedance and cutoff frequency
   .param w = 50 f = 1e6
2) Order n of the filter and reflection factor r in dB
   .param n = 4 r = 20
3) Functions for de-standardistion
   .func LL(x,y) (x/(2*pi*y))
   .func CC(x,y) (1/(2*pi*x*y))
4) Calculations for filters
   4a) auxilliary functions
       .func rho(y) (pow(10,-y/20))
       .func eta(x,y) = sinh((1/x)*asinh(1/rho(y)))
   4b) Standardised values of components
       g1: .func g1(x,y) (2*sin(pi/(2*x))/eta(x,y))
       g2: .func N2(x,y) (pow(eta(x,y),2)+pow(sin(1*pi/x),2))
            .func g12(x,y) (4*sin(1*pi/(2*x))*sin(3*pi/(2*x))/N2(x,y))
            .func g2(x,y) (g12(x,y)/g1(x,y))
       g3: .func N3(x,y) (pow(eta(x,y),2)+pow(sin(2*pi/x),2))
            .func g23(x,y) (4*sin(3*pi/(2*x))*sin(5*pi/(2*x))/N3(x,y))
            .func g3(x,y) (g23(x,y)/g2(x,y)
       g4: .func N4(x,y) (pow(eta(x,y),2)+pow(sin(3*pi/x),2))
            .func g34(x,y) (4*sin(5*pi/(2*x))*sin(7*pi/(2*x))/N4(x,y))
            .func g4(x,y) (g34(x,y)/g3(x,y))
5) Calculation or output resistance
   n = even:
       .func RL0(x,y) (x*pow((rho(y) + sqrt(1+pow(rho(y),2))),2))
       Call: RL0(w,r)
   n = odd:
       .func RL1(x) (x)
       Call: RL1(w)

Fig 6: Circuit diagram and script for 4th order Tschebyscheff low-pass filter.

help. The dialogue is closed with "OK" and the text is place on the list.

Now clicking "run" starts the simulation. When the simulation is complete the waveform viewer is opened automatically. The variables to be shown must be defined. To do this open "Plot Settings/Add Trace" and use the dialogue "Add Traces to plot". In this case these are S11 (v1) and S21 (v1), click both and complete by clicking "OK".

It is possible to change the scale of the axes. Pointing at one of the axes with the mouse changes the mouse pointer to a ruler. Click the mouse on the axes e.g. on the left axis in the picture opens the dialogue "Left Vertically Axis – Magnitude". The axis can be set to: linear, logarithmically or dB. The right axis shows the amplitude and phase delay of the envelope. The frequency axis can also be adjusted for a linear or logarithmic scale.

3.5. Results

Unfortunately the values of the components cannot be seen on the work space. They can however be seen using a small detour. First instruct LTspice to show an "extended listing" using the "control panel". Open "Tools/Control panel" and on the list of options select "operation" and activate "Generate Expanded listing". Start the simulation and with the schematic window still active, click on "View" in the menu and select "Spice error log", this opens the dialogue that shows the values of the components.

For the Butterworth low-pass filter with w = 50Ω, f = 1MHz, n = 4 the following values are found in the "Spice error log":

$$L1 = 6.09mH$$
$$C2 = 5.88nF$$
$$L3 = 14.70mH$$
$$C4 = 2.43nF$$

4.

Tschebyscheff low-pass filter

The circuit and the script for a 4th order Tschebyscheff low-pass filter is shown in Fig 6.

Apart from characteristic impedance, cut-off frequency and filter order the attenuation variation αD in the pass band is an important parameter for the design of Tschebyscheff filters. As an alternative to this the reflection factor ρ or better the return loss r can be used. A network analyser usually indicates the latter. The connection between these values is as follows:

$$\alpha_D = -10 \, \lg\left(1 - \rho^2\right)$$

$$\rho = \sqrt{1 - 10^{\frac{-\alpha_D}{10}}}$$

$$r = -20 \, \lg(\rho) = -20 \, \lg\left(|S_{11}|\right)$$

$$\rho = 10^{\frac{-r}{20}}$$

Example:

$$\alpha_D = 0.0436dB$$
$$\rho = 0.1$$
$$r = 20dB$$

4.1. Formulae for calculations

For the computation of Tschebyscheff low-pass filters the equations from [3] were used. As mentioned, the filter is defined by the characteristic impedance w, the cutoff frequency f, the filter order n and the return loss r. The following auxiliary functions clarify the equations used in the script.

$$rho = 10^{\frac{-2}{20}}$$

$$eta - \sinh\left(\frac{1}{n} a \sinh\left(\frac{1}{rho}\right)\right)$$
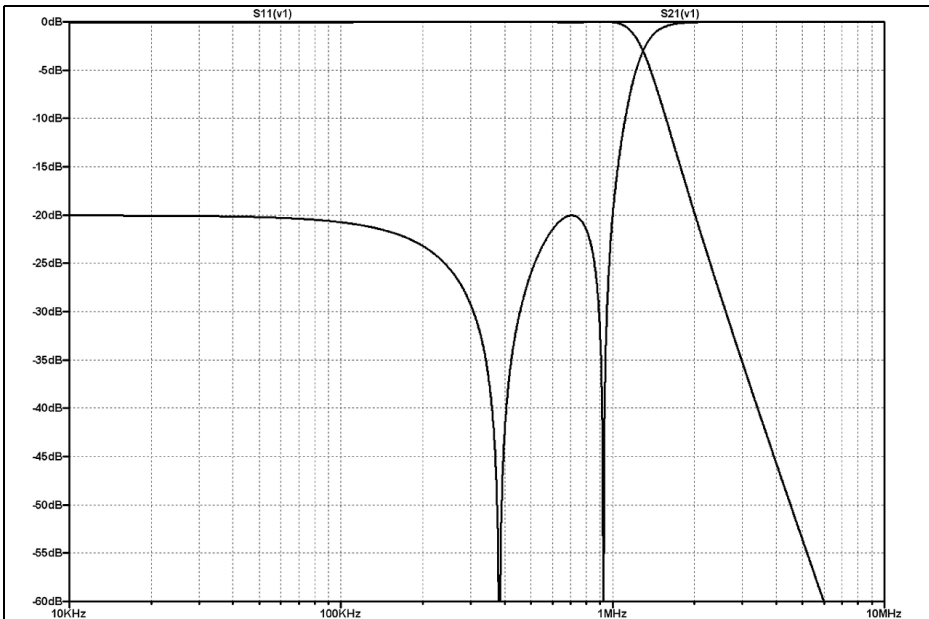
Fig 7: Frequency response of the Tschebyscheff filter.

$$g_1 = \frac{2\sin\dfrac{\pi}{2n}}{eta}$$

$$N_{k+1} = \sinh^2(eta) + \sin^2\left(\frac{k\pi}{n}\right)$$

$$g_k g_{k+1} = \frac{4\sin\left(\dfrac{2k-1}{2n}\pi\right)\sin\left(\dfrac{2k+1}{2n}\pi\right)}{N_{k+1}}$$

for k = 1… n-1

rho and eta are computed first. Afterwards the standardised value g1 of the first component is computed e.g.

g1 = 0.9314
 (for w = 50Ω, f = 1MHz,
 n = 4, r = 20dB).

The following components g2 to gk are computed iteratively by repeated application of the last two equations. Thus for k = 1, next N2 is computed. With this value the product (g1*g2) = 1.2033 can

be determined.

From this follows:
 g2 = 1.2033/0.9314 = 1.2919.

For k = 2 (g2*g3) is determined, and from it
 g3 = (g2*g3) /g2, etc. to k = n.

The de-standardisation takes place, as with the Butterworth filters by multiplication of inductances with LL (w, f) and capacities with CC (w, f).

4.2. Terminating resistance

For even filter orders n the output resistance of the filter depends on the reflection factor (rho). Since the filter is to be terminated at the output, the terminating resistance is equal to the output resistance of the filter. The standardised terminal resistance is computed as a function of the reflection factor from the following equation:

$$RL = \left(rho + \sqrt{1 = rho^2}\right)$$

20

1) Characteristic impedance and cutoff frequency
    .param w = 50 f = 1e6
2) Order n of the filter and reflection factor r in dB
    .param n = 4 r = 20
3) Functions for de-standardistion
    .func LL(x,y) (x/(2*pi*y))
    .func CC(x,y) (1/(2*pi*x*y))
4) Calculations for filters
4a) auxilliary functions
    .func rho(y) (pow(10,-y/20))
    .func eta(x,y) = sinh((1/x)*asinh(1/rho(y)))
4b) Standardised values of components
    g1: .func g1(x,y) (2*sin(pi/(2*x))/eta(x,y))
    g2: .func N2(x,y) (pow(eta(x,y),2)+pow(sin(1*pi/x),2))
        .func g12(x,y) (4*sin(1*pi/(2*x))*sin(3*pi/(2*x))/N2(x,y))
        .func g2(x,y) (g12(x,y)/g1(x,y))
    g3: .func N3(x,y) (pow(eta(x,y),2)+pow(sin(2*pi/x),2))
        .func g23(x,y) (4*sin(3*pi/(2*x))*sin(5*pi/(2*x))/N3(x,y))
        .func g3(x,y) (g23(x,y)/g2(x,y))
    g4: .func N4(x,y) (pow(eta(x,y),2)+pow(sin(3*pi/x),2))
        .func g34(x,y) (4*sin(5*pi/(2*x))*sin(7*pi/(2*x))/N4(x,y))
        .func g4(x,y) (g34(x,y)/g3(x,y))
5) Calculation or output resistance
    n = even:
        .func RL0(x,y) (x*pow((rho(y) + sqrt(1+pow(rho(y),2))),2))
        Call: RL0(w,r)
    n = odd:
        .func RL1(x) (x)
        Call: RL1(w)

Fig 8: Circuit diagram and script for the binary filter.

C1
{CC(w,f)*(1/g1(n,r))}

C3
{CC(w,f)*1/(g3(n,r))}

R2
{RL0(w,r)}

L2
{LL(w,f)*1/(g2(n,r))}

L4
{LL(w,f)*1/(g4(n,r))}

V1
AC 1 0
Rser={w}

.ac dec 1000 10k 10meg
.net I(R2) V1

1) Characteristic impedance and cutoff frequency
   .param w = 50 f = 1e6
2) Order n of the filter and reflection factor r in dB
   .param n = 4 r = 20
3) Functions for de-standardistion
   .func LL(x,y) (x/(2*pi*y))
   .func CC(x,y) (1/(2*pi*x*y))
4) Calculations for filters
4a) auxilliary functions
   .func rho(y) (pow(10,-y/20))
   .func eta(x,y) = sinh((1/x)*asinh(1/rho(y)))
4b) Standardised values of components
   g1: .func g1(x,y) (2*sin(pi/(2*x))/eta(x,y))
   g2: .func N2(x,y) (pow(eta(x,y),2)+pow(sin(1*pi/x),2))
       .func g12(x,y) (4*sin(1*pi/(2*x))*sin(3*pi/(2*x))/N2(x,y))
       .func g2(x,y) (g12(x,y)/g1(x,y))
   g3: .func N3(x,y) (pow(eta(x,y),2)+pow(sin(2*pi/x),2))
       .func g23(x,y) (4*sin(3*pi/(2*x))*sin(5*pi/(2*x))/N3(x,y))
       .func g3(x,y) (g23(x,y)/g2(x,y)
   g4: .func N4(x,y) (pow(eta(x,y),2)+pow(sin(3*pi/x),2))
       .func g34(x,y) (4*sin(5*pi/(2*x))*sin(7*pi/(2*x))/N4(x,y))
       .func g4(x,y) (g34(x,y)/g3(x,y))
5) Calculation or output resistance
   n = even:
       .func RL0(x,y) (x*pow((rho(y) + sqrt(1+pow(rho(y),2))),2))
       Call: RL0(w,r)
   n = odd:
       .func RL1(x) (x)
       Call: RL1(w)

Fig 9: Circuit diagram and script for 4th order Tschebyscheff high-pass filter.
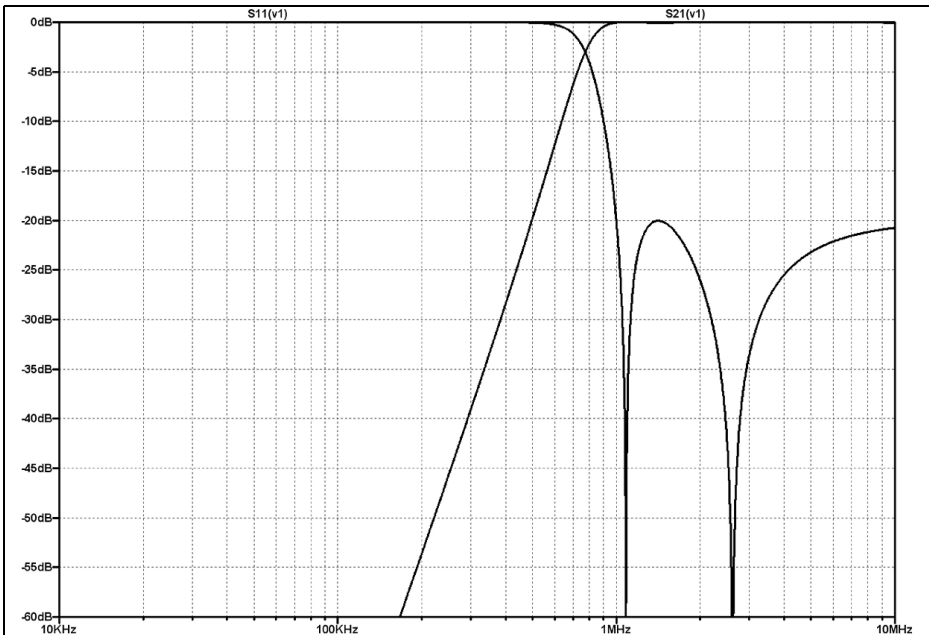
Fig 10: Frequency response of the high-pass filter.

For odd filter orders the terminating resistance is equal to the characteristic impedance.

RL=1

The de-standardisation takes place by multiplication with the characteristic impedance w. The generator resistance is always equal to the characteristic impedance.

4.3. Programming and results

The process to input the circuit and script for the Tschebyscheff filter is the same as for the preceding Butterworth filter. The calculation of the standardised components uses the functions shown in 4.1.

The components L1, C2, etc. call the functions LL () or CC () and gk (). For gk () the following mathematical functions are used: sin (), pow (x, y) (Power x height y) and the user-defined function Nk (). From Nk () the function eta () is calculated and from this rho (). The

parameters x and y are passed on to all functions.

The frequency response of the filter is shown in Fig 7. The values of the components taken from the SPICE error log are:

$$
\begin{array}{ll}
L1 = & 7.41\text{mH} \\
C2 = & 4.11\text{nF} \\
L3 = & 12.55\text{mH} \\
C4 = & 2.42\text{nF}
\end{array}
$$

The terminating resistance RL is:

$$RL = \quad 61.04\Omega.$$

5.

Binary low-pass filters

To design a binary low-pass filter the circuit diagram must be changed as

shown in Fig 8. The new component layout means that the standardised values must be recalculated: g1, g2, and so on and link c1 with g1, L2 with g2, C3 with g3 and L4 with g4.

In addition a small change to the script is required. The calculation of the terminal resistance RL0 () for binary filters the sign of the exponent of the bracketed term must be negative:

$$RL = \left(rho + \sqrt{1 + rho^2}\right)^2$$

6.

High-pass filter

High-pass filters are realised from a reference low-pass filter using frequency transformation [4].

The components of a high-pass filter are realised by replacing each inductor in the reference low-pass filter by a capacitor and each capacitor by an inductor. The circuit in Fig 9 was developed in this way from the circuit in Fig 6.

The standardised values of the components apply both to the Tschebyscheff and Butterworth filters:

$$g_{k,HP} = \frac{1}{g_{k,TP}}$$

It can be seen from Fig 9 that the script does not change from the low-pass filter. The only change concerns the value assignment to the components; the reciprocal values are used for gk. The frequency response of the filter is shown in Fig 10.

7.

Standardised values for the components

So far to de-standardised values the components have be used. These values can be found in the SPICE error log. If you are interested in the standardised values the scaling of the components with LL (w, f) and CC (w, f) must be prevented. This can be achieved with a small adjustment to make the specification of the filter parameters specify standardised values.

So the standardised characteristic impedance must be w = 1 and the standardised cutoff frequency ω = 1, and

$$f = \frac{1}{2\pi} = 0.1592$$

No other changes to the script are required. The generator resistance Rser = {w} is automatically set to 1, also the values also LL (w, f) is = 1 and CC (w, f) = 1.

The frequency range of the "AC" instruction must be changed on the standardised frequency, e.g. "AC 1000 0.01 100".

8.

Summary

After a very short introduction to the SPICE simulator LTspice/SwCAD3 from Linear Technology a method was described to calculate Butterworth and Tschebysheff filters using LTspice.

The standardised and de-standardised values of the components for low and high-pass filters and binary circuits were calculated.

The network analysis in LTspice makes it possible to show the results as S parameters.

LTspice/SwCAD3 is a reliable aid that becomes more useful as you become familiar with its operation.

9.

Literature

[1] Berkeley University OF California; U.S.A
http://www.coe.berkeley.edu/labnotes/0502/history.html
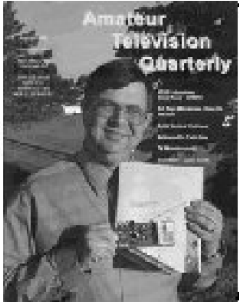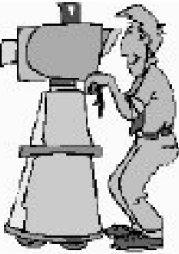
[2] Linear Technology;
http:// www.linear.com

[2A] Download site:
http://ltspice.linear.com/software/swcadiii.exe

[2B] Download site for help:
http://LTspice.linear.com/software/scad3.pdf

[3] D. Budimir, Generalized filter Design by computer Optimization, Artech House, 1998

[4] R. Saal, Manual to the filter design, Hüthig, 1988

Henning Christof Weddig, DC5LV

The AGC module

An introduction to an IF amplifier for amplitude modulated signals was given in [1]. The gain must be controlled depending on the amplitude of the received signal taken from the output signal of the IF amplifier. In addition an external coarse control is required this is described in the following article.

1.

Introduction

The gain of the IF amplifier described in [1] is controlled by 84dB with the voltage V_{AGC} ranging from 2.0V to 2.84V generated by the output signal of the IF amplifier.

Alternatively the control voltage could be taken from a potentiometer on the front panel of the receiver to change the gain by hand. A mixed manual and automatic gain control should be possible. These options are realised in this module.

AGC is the abbreviation of the term "Automatic Gain Control", while the gain controlled by hand is the MGC (Manual Gain Control).

The requirements of the receiver controls are to be described briefly below:

- The maximum gain of the IF amplifier and the automatic gain control should be designed in such a way that the smallest signal for SSB (B = 2.7kHz) produces the nominal output level. The automatic control starts from the minimum operating sensitivity (-107dBm) and keeps the output signal constant at the pre-defined level (proportional control).

- The attack time should be short, <5ms, so that over regulation is kept to a minimum and reduces annoying "plops" in the loudspeaker.

- When there is a pause in the speech of an SSB signal it can cause a rushing sound as the gain control changes. The hold time should have two selectable times: AGC Short and AGC Long. The hold time for AGC Short should be approximately 400ms and for AGC Long it should be approximately 1 to 2s, after that period there should be a fast decay time of approximately 5 to 10ms. This kind of control is called "hanging regulation".

- The AGC module should have sufficiently good control so that fast

changes in the signal level do not affect the control characteristics unfavourably.

- For some operating modes e.g. CW, it is desirable for the gain to be set manually.

- A mixed manual/automatic control should be available. In this mode the basic gain should be set manually using the MGC potentiometer on the front panel. If the received signal goes over a given threshold the automatic control should take over.

- Short, one off input signal changes (crackling noises etc.) in AGC Long mode should be compensated but not for an unnecessarily long time (preservation time).

- The receiver should be muted by a special input signal.

- If the operator changes from manual to automatic AGC mode when the manual gain is set to a high level, the changeover can cause the receiver to be blocked temporarily. This is caused by the capacitor in the control circuit storing charge. This can be discharged by a monostable triggered when the AGC mode is changed.

- Muting the receiver in transmit mode can be achieved by a high level input to the IF amplifier. When switching back to receive the IF amplifier gain has to be adjusted back to the level associated with the received signal of the remote station, this can cause annoying "plops". This can be overcome by storing the old control level of the AGC while in transmit mode.

1.1 Generating the control voltage

The control voltage can be generated from the demodulated signal (AF) or the IF signal. With the first method the control can only follow an input signal level change at the rise time of the audio frequency signal. If very low frequency audio signals, e.g. signals of 300Hz (the lower cutoff frequency) are demodulated, the rise time is that of the 300Hz:

$$T = 1/f = 1/300 = 3.3ms$$

If the control capacitor is charged so fast that it follows the sinusoidal curve without delay, the rise time of the control voltage would be approximately a quarter of this time = 0.825ms. This time would surely be sufficient which is contrary to the opinion expressed in technical journals, e.g. [2]. However the rise time is a function of the demodulated signal frequency and that is not acceptable. Therefore the chosen method is to generate the control voltage from the IF signal. With this method the BFO signal must be avoided so that it does not limit the range of control.

In the 1970s Plessey produced some interesting integrated circuits for IF amplifiers, unfortunately these are no longer available. Among other things the control level circuit, SL621, gave the ability to compensate for short impulse noises without muting the receiver for a long time. This function is realised using operational amplifiers.

2.

Block diagram of the AGC module

The block diagram for the AGC module is shown in Fig 1.

After the author had finished building the AGC module and connected it to the IF amplifier described in [1] he had a big surprise: without any RF input the control voltage was 2.2V.

The reason is the broadband noise that the IF amplifier generates. This noise is
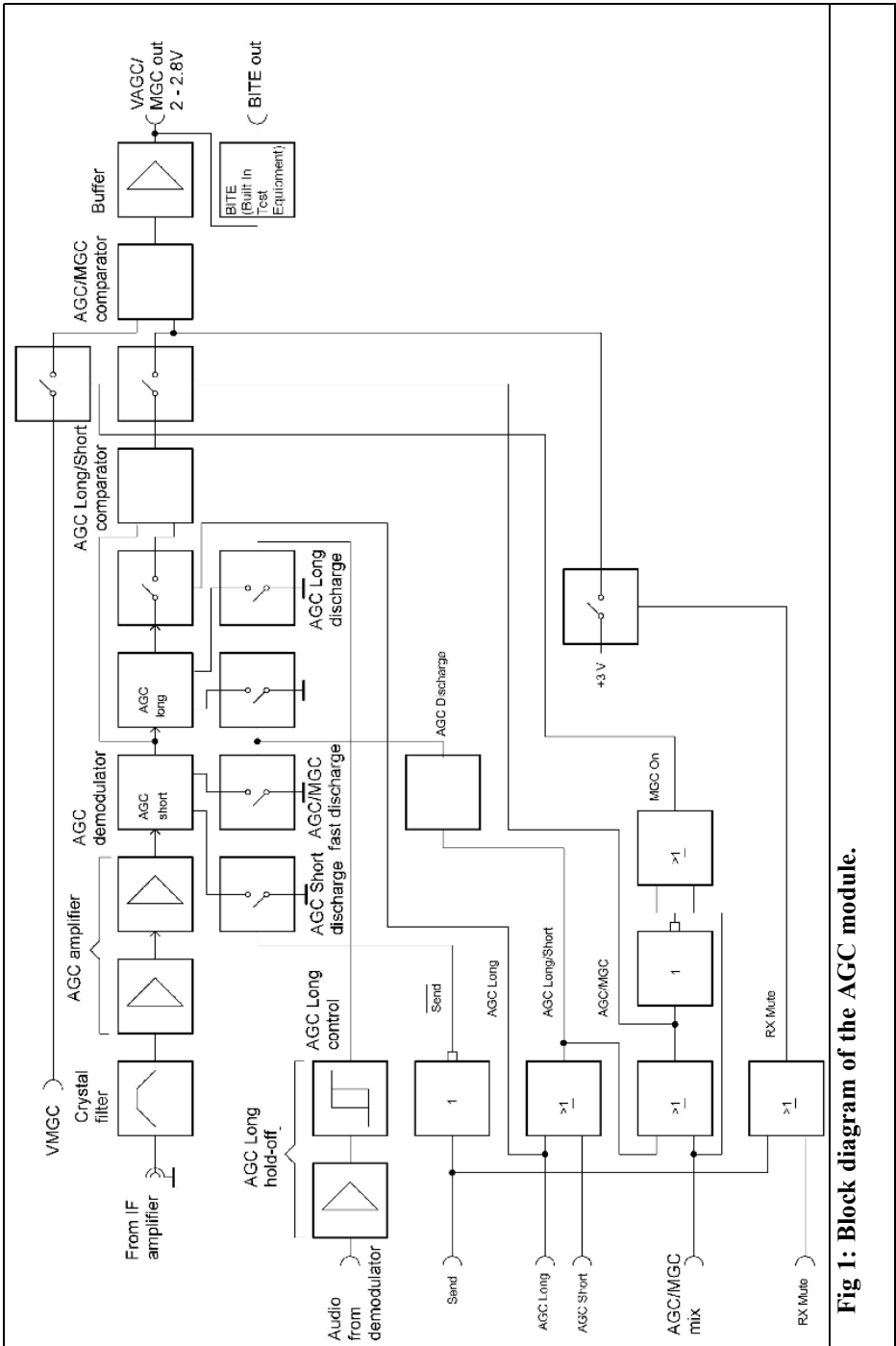
Fig 1: Block diagram of the AGC module.

generated in the bandwidth of the band-pass filter between the two AD603 amplifiers and the single tuned cicuit at the output of the buffer stages, this is approximately 1MHz, see Fig 11 in [1]. A signal-to-noise ratio of 20dB was measured using a spectrum analyser with a nominal input signal, however the resolution bandwidth (RBE) of the measuring instrument was set to 3kHz. The AGC module "sees" the higher amplitude wide-band noise produced by the IF amplifier that it converts into a control signal.

By using a crystal filter (XF910 from KVG), matched to 50Ω using L matches, between the IF amplifier and the AGC module the control voltage dropped to under 1V with no RF input signal.

The nominal output signal (-30dBm = 7.08mVeff = 20mVpp) of the IF amplifier is too small for direct rectification; therefore a two stage amplifier is used before the diode so that the output is a proportional DC voltage. An operational amplifier buffer is used to charge a capacitor via a resistor with a time constant shown as AGC Short in the block diagram. The capacitor voltage is fed to one half of the AGC Long/Short comparator and also charges a larger capacitor for AGC Long. This AGC Long voltage is buffered by another operational amplifier and fed to the other half of the comparator via an analogue switch. This switch is only closed in the AGC Long position. The higher of these two voltages is transferred to the output of the comparator. If a brief impulse is received in the AGC Long position it is compensated by the AGC Short regulation with a relatively short rise time that does not have a long-term affect on the receiver output.

The output signal from the AGC comparator is fed via an analogue switch to the AGC/MGC comparator. The second input of this comparator is connected via

an analogue switch to the V_{MGC} input, which is the MGC potentiometer on the front panel of the receiver. In the AGC/MGC mixing mode both of these analogue switches are closed so that the higher voltage takes over the gain control. This prevents the receiver being overridden by a low manual control voltage when there is an RF signal that is too high. The control voltage is fed to the IF amplifier and signal strength meter via a buffer.

The "MUTE" mode connects a high control voltage of +3V to the comparator input via an analogue switch that sets the IF amplifier to its lowest gain and thus is practically switched off.

If the receiver is going to be operated with a transmitter, the control voltage is stored in the AGC Short capacitor before the changeover from receive to transmit (Transmit signal goes high). The ground connection of the discharge resistor is broken using an analogue switch so that the capacitor cannot self-discharge. The "MUTE" analogue switch is also closed in transmit mode.

In AGC Long mode the control voltage is maintained for 1 to 2s during pauses in speech. In addition the demodulated AF signal module of the demodulator is increased using a capacitor with a relatively short time constant.

The discharge time constant is much longer. The capacitor voltage controls a schmitt trigger whose output switches an analogue switch to discharge the AGC Long reservoir capacitor via a low value resistor to give a short discharge time. Thus the control of the receiver is quickly returned after the hold time expires.

When switching from MGC to AGC Long or AGC Short a monostable is triggered that discharges the AGC Long and AGC Short reservoir capacitors
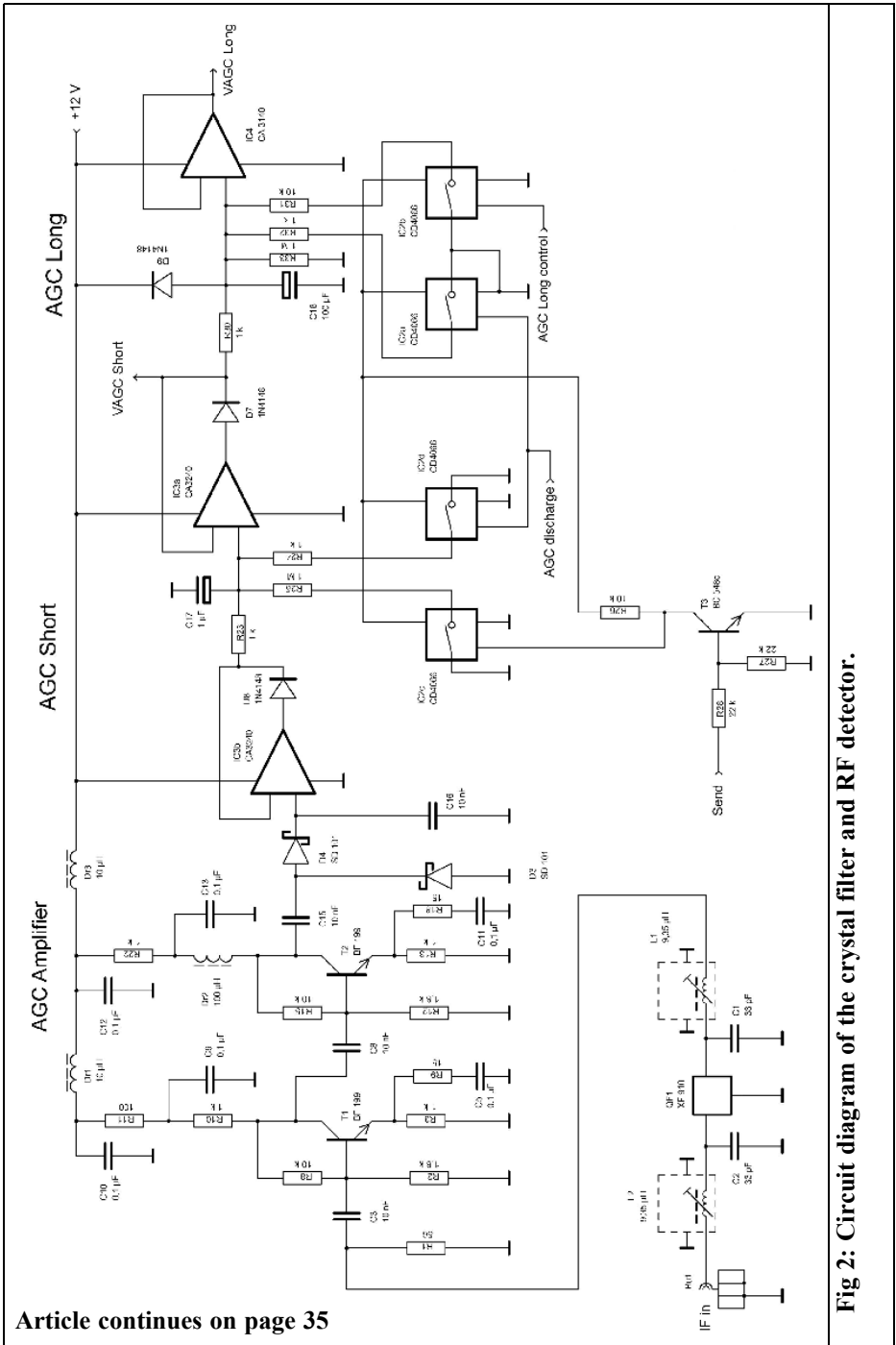
Fig 2: Circuit diagram of the crystal filter and RF detector.

Article continues on page 35

Index of Volume 39 (2007)

A complete index for VHF Communications Magazine from 1969 to the current issue is available on the VHF Communications Magazine web site - http://www.vhfcomm.co.uk. The index can be searched on line or downloaded in pdf of Excel format so that it can be printed or searched on your own PC. If you are not connected to The Internet you can write to or fax K.M. publications for a printed copy of the index which will cost £2.50 plus postage.

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

Fax: +44 (0)1582 581051

Email: andy@vhfcomm.co.uk

VHF Communications Back Issues Available

Year	Q1	Q2	Q3	Q4
2007	Y	Y	Y	Y
2006	Y	Y	Y	Y
2005	Y	Y	Y	Y
2004	Y	Y	Y	Y
2003	Y	Y	Y	Y
2002	Y	Y	Y	Y
2001	Y	Y	Y	Y
2000	Y	Y	Y	Y
1999	Y	Y	Y	P
1998	Y	Y	Y	Y
1997	Y	Y	Y	Y
1996	Y	Y	Y	Y
1995	Y	Y	Y	Y
1994	Y	Y	Y	Y
1993	Y	P	P	P
1992	Y	Y	Y	Y
1991	P	P	P	P
1990	P	P	P	P
1989	P	P	P	P
1988	P	P	P	P
1987	P	P	P	P
1986	P	Y	Y	Y
1985	Y	Y	Y	Y
1984	Y	Y	Y	Y
1983	P	P	Y	Y
1982	P	P	P	P
1981	P	L	P	P
1980	P	P	P	P
1979	L	P	P	P
1978	P	P	P	P
1977	P	P	P	P
1976	P	P	P	P
1975	P	P	P	P
1974	L	P	P	P
1973	P	P	P	P
1972	P	P	P	P
1971	P	P	P	P
1970	P	P	P	P
1969	P	P	P	P

Key: Y = Available   L = Low quantity   P = Photocopy
The back issue sets contain issues marked Y. Issues marked L or P must be ordered separately.

- Single issues from 1969 to 2006, £1.00 each plus postage

- Single issues from 2007 volume, £4.70 each plus postage

- Complete 2007 volume, £18.60 plus postage

- Back issue set up to 2006 (69 magazines), £45.00 plus postage

- Back issue set up to 2007 (73 magazines), £65.00 plus postage

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

Tel / Fax +44 (0)1582 581051   Web site http://www.vhfcomm.co.uk

quickly with two analogue switches, so the receiver is not blocked by a high control voltage.

A comparator is used to monitor the control voltage when a test signal is applied to the receiver input. If the control voltage is correct the output is 0V indicating correct operation of the AGC module. This is a BITE (Built In Test Equipment) function.

3.

Circuit description

For better legibility of the complete circuit diagram of the AGC module is divided into three separate diagrams, see Fig 2, 3 and 4.

3.1. Crystal filter

The crystal filters, AGC amplifiers, RF rectification plus the AGC Short and Long circuits are shown in Fig 2. The output signal of the IF amplifier is fed to socket Bu1. Paragraph 2 explained how broadband noise from the IF amplifier causes an unwanted high control voltage. To reduce the bandwidth a crystal filter type XF910 from KVG is used. This is matched to 50Ω with L matches (L2, C2 and L1, C1). The values of the matching components can be computed with the Freeware program "RFSim99". Alternatively they can be calculated with the following formulae (see [5] for their derivation):

$$X_1 = \sqrt{R_G(R_F - R_G)} \qquad X_2 = \frac{R_F}{\sqrt{\frac{R_F}{R_G} - 1}}$$

$$L_1 = \frac{X_1}{\omega_0} \qquad C_1 = \frac{1}{\omega_0 X_2}$$

Where:

RG = generator resistance (50Ω)
RF = filter resistance (for the XF910 = 6kΩ)
ω_0 = centre frequency of the filter

For the XF910 filter the following values result:

$$X_1 = \sqrt{50\Omega(6000\Omega - 50\Omega)} = 524\Omega$$

$$L_1 = \frac{524\Omega}{6.28 \cdot 9 \cdot 10^6 \frac{1}{s}} = 9.26 \cdot 10^{-9} \frac{\frac{V}{A}}{\frac{1}{s}} = 9.26\,\mu H$$

$$X_2 = \frac{6000\Omega}{\sqrt{\frac{6000\Omega}{50\Omega} - 1}} = 550\Omega$$

$$C_1 = \frac{1}{6.28 \cdot 9 \cdot 10^6 \frac{1}{s} \cdot 550 \frac{V}{A}} = 32.15\,pF$$

The inductor was wound on a 7mm former (7F1S) and an F10b core from Neosid.

The number of turns required can be calculated as follows:

$$L = A_L n^2 \rightarrow n = \sqrt{\frac{L}{A_L}}$$

NEOSID give 23nH/turn for the F10b material. The number of turns for an inductance of 9.26μH is given by:

$$n = \sqrt{\frac{9260nH}{23nH}} = 20\,turns$$

The author has discovered that KVG no longer manufacture this filter, however 9MHz (9M7A) and 10.7MHz filters are available from [3]. The author has not tried these filters. The additional insertion loss of approximately 4.5dB must be

considered because the output level of the IF amplifier and demodulator modules must be increased by this amount.

3.2. AGC amplifier and AF demodulation

The input impedance of the amplifier following the crystal filter is much higher than the system impedance of 50Ω. The correct impedance is achieved with R1.

To roughly determine the gain required for the AGC amplifier with a nominal –30dBm input for an AGC voltage of 2V the detector diodes must be investigated. The Schottky diodes D3 and D4 form a voltage doubler. So the voltage across C15 is given by:

$$V_{OUT} = V_{AGC} + 2 \times V_{DIODE}$$
$$= 2 + 2 \times 0.2$$
$$= 2.4V$$

The voltage across capacitor C16 is the peak voltage less a small voltage of 0.2V dropped across diode D4. An input of -30dBm corresponds to an effective voltage of 7.08mV. The peak-to-peak voltage is:

$$V_{pp} = 7.06 mV_{eff} \cdot 2 \cdot \sqrt{2} = 20 mV_{pp}$$

The output voltage divided by the input voltage gives the voltage gain, V_U, required:

$$V_U = \frac{V_{OUT}}{V_{IN}} = \frac{2400 mV}{20 mV} = 120$$

$$V_U(dB) = 20 \cdot \log V_U = 20 \cdot \log 120 = 41.4 dB$$

This high gain is achieved using a two stage RC coupled amplifier comprising of T1 and T2 with negative feedback. Choke Dr2 provides good performance for signals with level changes.

Originally, normal silicon diodes (1N4148) were used following the two-

stage transistor amplifier but these were not acceptable. Schottky diodes have a lower forward bias voltage of 0.2V and provide a higher output voltage. The diode type shown in the circuit diagram is SD101 by almost any small signal Schottky diode will work e.g. HP 2800.

3.3. AGC Short

The rise time of the two-stage AGC amplifier was measured as 10ms instead of the 1ms expected. The high internal resistance of the T2 amplifier stage caused this discrepancy between the calculated and measured values. Therefore the output is buffered with an operational amplifier IC3b. According to data sheet [6] the output resistance is typically 60Ω. D8 prevents the capacitor C16 from discharging through the operational amplifier if the input voltage across C15 is lower than that across C17.

R23 and C17 form the rise time constant for positive input increases and R25 discharges the tantalum electrolytic capacitor C17 for the decay time constant. The CMOS operational amplifier IC3a buffers the voltage across C17. The rise time constant for AGC Short is given by:

$$\tau_{risetime} = R \cdot C = 1k\Omega \cdot 1\mu F = 1ms$$

and the decay time constant is given by:

$$\tau_{falltime} = R \cdot C = 1M\Omega \cdot 1\mu F = 1s$$

Note:

The rise time applies only to the open loop control circuit. If the loop is closed with the IF amplifier this time is reduced. This must be dealt with depending on the IF amplifier used.

During the changeover to transmit mode (St3 "send" goes high) the control voltage is stored in the AGC Short capacitor by disconnecting the discharge resistor R29 from ground with the analogue
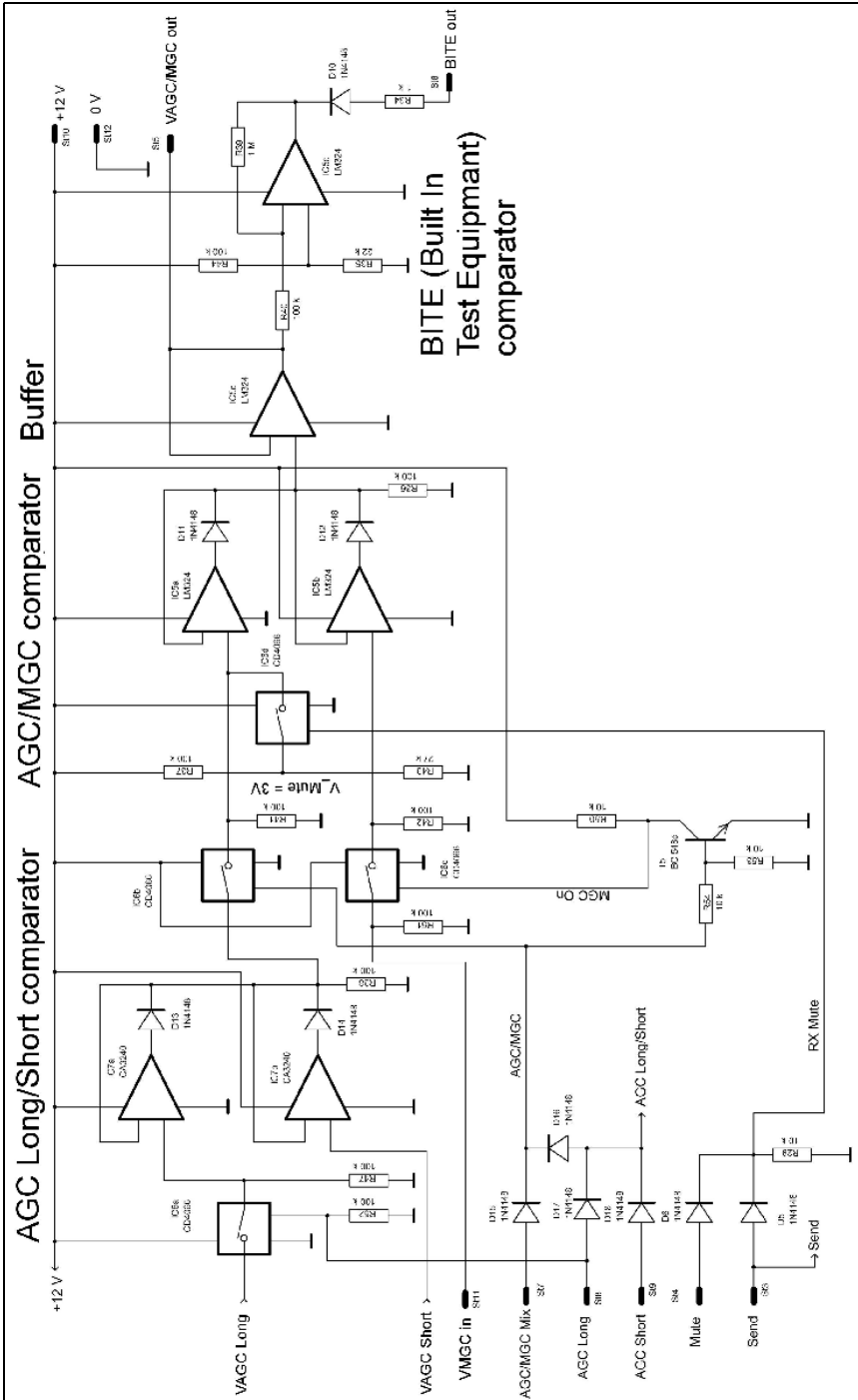
Fig 3: Circuit diagram of the comparators.

switch IC2c controlled by T3.

The discharge resistor R24 is switched to ground with IC2d when changing from manual control (MGC) to AGC Long or AGC Short, which activates the monostable IC8. This prevents the receiver being blocked by a possible high control voltage being applied.

3.4. AGC Long

The output voltage of the buffer amplifier IC3a is fed to diode D7, resistor R30 and C18 that form a peak detector. The operation amplifier IC4 buffers the capacitor voltage. The rise time is determined by R30 and C18.

The prototype showed a small leakage current flows from the inverting input of the CMOS operational amplifier IC4 causing a slow additional charge for C18. R33 defines the hold time for the control voltage during speech pauses in the received signal.

The discharge resistor R32 is connected to ground by the analogue switch IC2a when switching from manual control (MGC) to AGC Long or AGC Short (monostable IC8 triggered), this quickly discharges the AGC Long capacitor. The diode D8 discharges the capacitor when the supply voltage is switched off and prevents the stored energy flowing through the input protection diodes of the buffer amplifier IC4 thus preventing any damage to the IC.

3.5. AGC Long/Short comparator

The circuits of the AGC Long/Short comparator and AGC/MGC comparator are shown in Fig 3.

The comparator circuit formed by the two CMOS operational amplifiers in IC7 passes on the higher of the two control voltages to the output. The AGC Long control voltage from the buffer amplifier IC4 is fed to the non-inverting input of

IC7a via the analogue switch IC6a. R47 discharges the leakage current when the switch is open.

The AGC Short control voltage is constantly fed to the non-inverting input of IC7b.

The outputs of the operational amplifiers are connected by the diodes D13 and D14, R38 provides a DC patch to discharge any leakage currents that may affect the function of the comparator. Because the diodes are inside the control loop of the operational amplifiers their temperature dependant characteristics do not affect the output

3.6. AGC/MGC comparator

For manual gain control the analogue switch IC6b switches off the internally generated control voltage and the MGC potentiometer is connected to the AGC/MGC comparator, comprising IC5a and IC5b, via the analogue switch IC6c. The circuit is identically to the AGC comparator.

Mixed AGC/MGC is also possible, in this case the higher of the control voltages (MGC or AGC) are fed to the IF amplifier. In this mode the analogue switch IC6b is closed.

3.7. Buffer stage

The output of the AGC/MGC comparator is connected to the buffer amplifier IC5c. The output is connected to St5, which is the control voltage fed to the AGC input of the IF amplifier. A signal strength meter can also be connected here.

3.8. BITE (Built In Test Equipment)

The complete receiver has a self-check mechanism (BITE), when a know test signal is fed into the antenna input. The correct function of the AGC module is checked with the comparator IC5d. The reference voltage (approximately 2.2V)
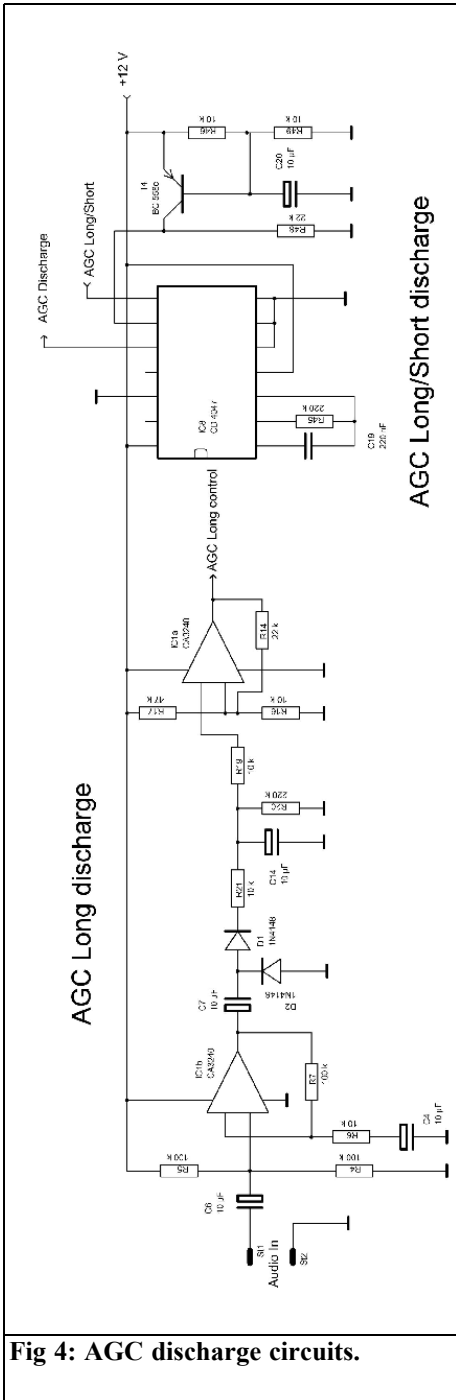
Fig 4: AGC discharge circuits.

is determined by the voltage divider R44/R35. R41 provides a small hysteresis. If the control voltage exceeds the reference value the output of the operational amplifier is 0V connected to St6 via D9 and R34.

3.9. Logic

The diodes D5, D6, D15 - D18 are linked to control the inputs: (St3) send; (St4) Mute; (St7) AGC/MGC mix; (St8) AGC Long; (St9) AGC Short. They control the appropriate analogue switches.

In transmit mode, the control signals at St9, St8 and St7 must be 0V to prevent back supply of the signals of the analogue switches IC6d and IC6b.

3.10. AGC Long/Short discharge

The AGC Long/Short discharge and AGC Long discharging circuits are shown in Fig 4.

In order to prevent the receiver being muted during the switchover from MGC mode to AGC Long or AGC Short mode the monostable IC8 is triggered, on pin 8, by the signal AGC Long/Short (see fig. 3). T4 is used to reset the monostable when the supply is switch on. The length of the monostable pulse is set by R45/C19 and is approximately 200ms.

The output signal (AGC discharge) feeds the two analogue switches (IC2d and IC2a, see fig. 2). They connect resistors R24 and R32 to ground to quickly discharge the capacitors; AGC Short (C17) and AGC Long (C18).

3.11. AGC Long discharge

The demodulated AF signal is connected to ports St1 and St2 and then via C6 to the non-inverting input of the operational amplifier IC1b. R4 and R5 set the inverting input at half supply voltage because IC1 uses a single voltage supply. The gain of IC1b is set by R6 and R7:
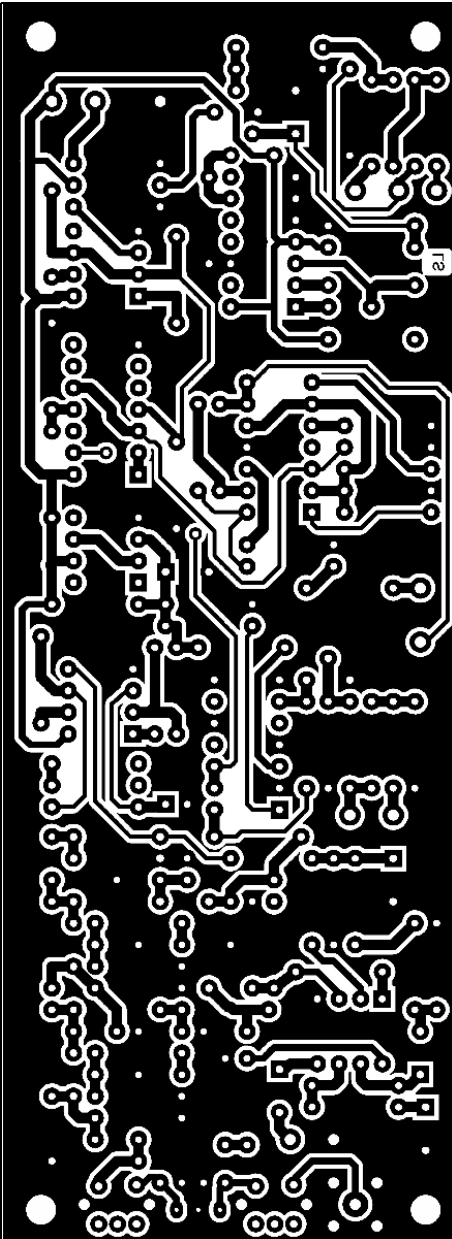
Fig 5: PCB layout for the bottom side of the AGC module.
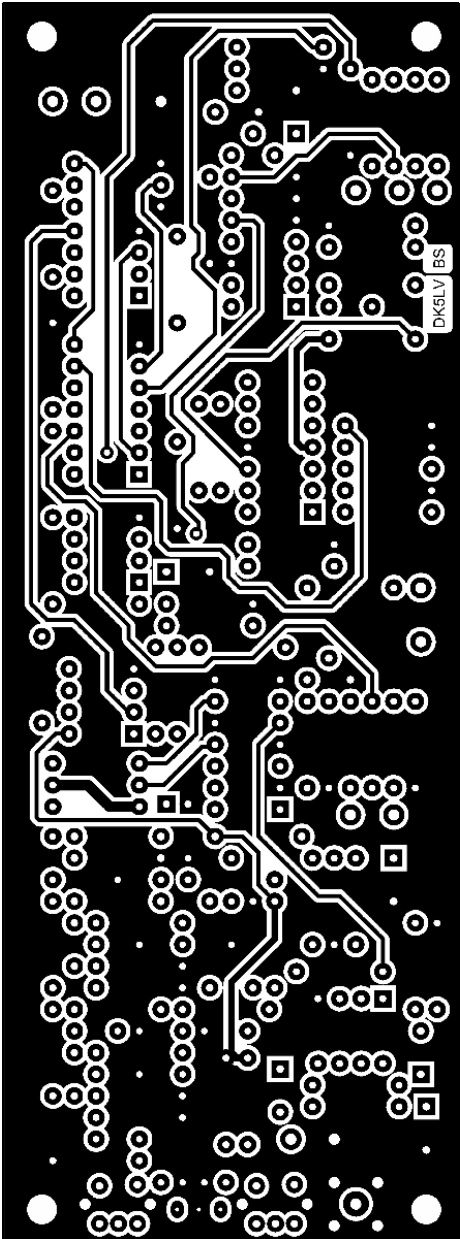
Fig 6: PCB layout for the component side of the AGC module.

Fig 7: Component layout for the AGC module.

$G = 1 + R7/R6 = 1 + 100/10 = 11$

Capacitor C4 keeps the DC voltage at both inputs of the operational amplifier equal. The amplifier feeds a voltage doubling detector circuit made from C7, D1 and D2. The DC voltage generated is fed via R21 to the reservoir capacitor C14 and discharge resistor R20. The schmitt trigger circuit formed by IC1b and R14, R16 and R17 that provide hysteresis feeds the analogue switch IC2b. This connects R33 to ground to quickly discharge the AGC Long reservoir capacitor.

4.

Circuit design and alignment

The AGC module is built on a 147mm x 55mm double sided PCB using conventional wired components. Figs 5 and 6 show the PCB layout and Fig 7 shows the component layout.

To test the circuit, connect a -30dBm RF signal in the range 8 - 10MHz (The amplifier is relatively wide band) to Bu1. The DC voltage on the cathode of D4 should be $2V \pm 0.1V$. Any deviations from this can be adjusted by changing R7 or R21.

Using a digital voltmeter, measure the DC voltage on the positive terminal of C15 and pin 1 of IC3a, both measurements should be the same.

Connect St7 (AGC Short) to +12V. Follow the signal path and the control voltage should also be found at the junction of R13, D12, D13; D10, D11, R36; Pin 8 of IC5c and St5.

Connect St8 (AGC Long) to +12V, St7 to 0V, and an audio signal (1kHz, 1Vpp) to St1, the control voltage will be on pin

3 of IC7a. Pin 5 of IC2b should be at logic 0. If the audio signal is switched off, pin 5 should change to logic 1 after approximately 1 to 2s and the voltage on pin 3 should fall to approximately 200mV.

Connect St9 (AGC/MGC mix) to +12V, St7 plus St8 to 0V and a potentiometer to provide a variable voltage on St11 (V_{MGC}), the voltage on St5 can be adjusted higher than voltage from the RF input.

If St7 or St8 are connected to +12V the monostable, IC8, will generate a short pulse in both cases.

Connect St4 (send) to +12V, T3 should switch on, the control voltage can be measured on pin 9 of IC2c. The voltage at St5 should be approximately 3V, the exact voltage depends on the values of the voltage divider R37, R43.

To be continued.

6.

Literature

[1] An IF amplifier with "dB linear gain control, Henning Christof Weddig DK5LV, VHF Communications Magazine 3/2007, pp 158 - 177

[2] Ein ZF-Nachsetzer, wie ihn die Industrie nicht Baut, Werner Schnorrenberg, DC4KU, Funkamateur, 10, 11, 12 /2000 and 1/2001

[3] Single port filter 9M7A, Funkamateur online Shop,
https://ssl.webpack.de/box73.de/catalog/index.php?
cPath=82_100&osCsid=8b28a1aecfc55e69615d3409472e75cb

[4] Data sheet AD603, revision G, Analog Devices 2005

[5] Technical Introduction Filters/Discriminators, KVG GmbH

[6] Data sheet CA3240, Intersil

John Fielding   ZS5JF

A New Band Pass Filter Design for µWave Projects

1.0

Introduction

Many microwave construction projects require narrow band pass filters for various parts of the system. These may be filters for frequency multiplier chains and receiver front end filtering. Whereas there have been many different designs published over the years, all of which work satisfactorily when constructed correctly, most designs require a fairly high level of skill in the machining and construction. The traditional resonator filters are split into two similar but quite different design concepts. The "Comb-Line" filter is named because the resonator rods are like the teeth of a comb and are all grounded at the same end. By contrast the "Inter-Digital" filter transposes the grounded end for every other rod. For both of these filter types the spacing between adjacent resonator rods or "fingers", lengths and diameter of the rods and the tapping up from the grounded end for the input and output connectors is critical. Any errors can result in poor impedance matching, excessive ripple in the pass band, excessive insertion loss and poor attenuation in the stop bands.

The filter to be described is similar to the two traditional filters but avoids the
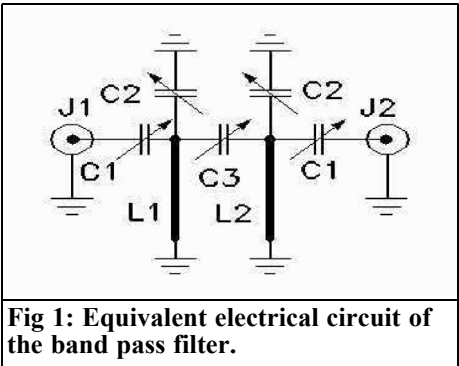
Fig 1: Equivalent electrical circuit of the band pass filter.

problems of accurate rod spacing and length. This is achieved by using two critically coupled cavity resonators with adjustable top coupling and impedance matching to 50Ω. The equivalent electrical circuit is shown in Fig 1.

Connectors J1 & J2 are SMA panel mount connectors with a long stub nose, which have the extended centre pin surrounded by PTFE. These are made by several manufacturers and are designed to allow the SMA bulkhead connector to be bolted to a wall of considerable thickness and the hole bored for the centre pin forms a 50Ω coaxial transmission line with the PTFE as the dielectric. These are available in various standard lengths; the longest being 0.59″ (15mm) and the shortest is 0.16″ (4mm). The centre pin protrudes a few millimeters proud of the PTFE. The diameter of the
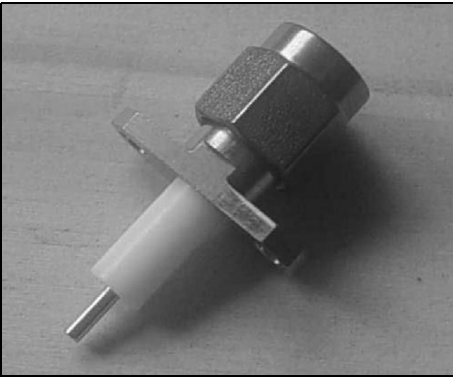
Fig 2: SMA connector with long centre pin and PTFE dielectric.

PTFE dielectric is nominally 4.1mm and a hole of 4.2mm bored in the equipment wall allows a push fit, with constant 50Ω impedance. A typical SMA stub nose connector is shown in Fig 2. They are available with male or female ends to suit the application.

Tuning capacitors C1 & C2 are formed by the SMA centre pin and the resonator rod in a "coaxial capacitor". Capacitor C3 is the top coupling capacitor and is formed by the fringing capacity between the top of the first resonator rod and the second resonator rod, each in their own shielded cavity. This capacitor needs to be an extremely low value, typically less than 0.05pf. To obtain the very small value of capacity required the two resonator rods to "see" one another by a small iris cutout between the adjacent cavities. The aperture of the iris can be controlled by the coupling screw, which partially blocks the iris. This is shown in the general arrangement drawing, which is a side view of the filter. Fig 3 shows the mechanical assembly of the 2-resonator filter.

The body of the filter is made from a block of aluminium 30mm x 50mm in profile and the height of the block (P) is determined by the operating frequency. The exact dimensions of the outer surfaces are not critical. Although 30mm x 50mm is quoted the piece of aluminium used by the writer is actually 31.75mm x 50.8mm, this being an imperial size of 1.25″ x 2″. Most metal wholesalers sell metal by weight and it is not uncommon to find that sizes are rounded up or down to the nearest whole number in the metric measurement system. This is because the cost of a new extrusion die to form the various sections is very expensive and rather than make a new die to suit the metric measurement system they round the size when converted to metric dimensions. Often when a piece of metal bar or sheet is measured it turns out it is in fact an imperial size in disguise!
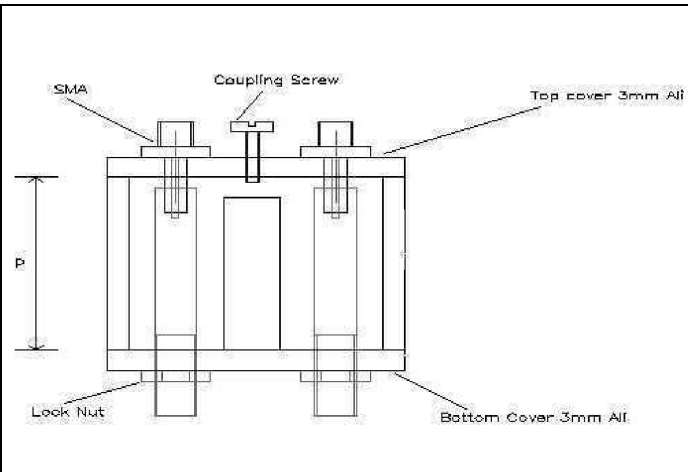
Fig 3: Side view of the assembled band pass filter.
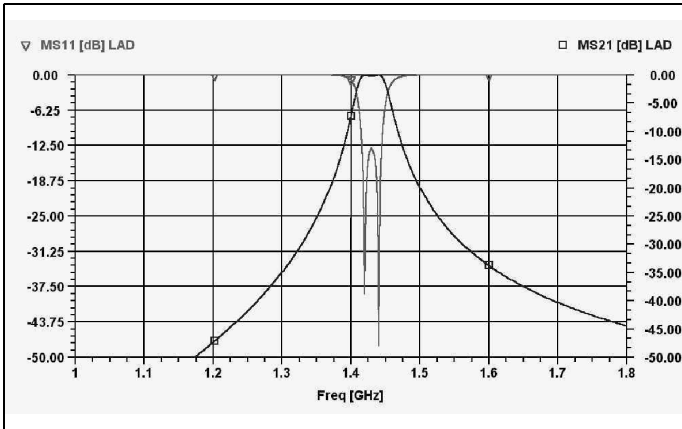
Fig 4: Simulated response plot for the 2-resonator filter at 1420MHz.

The filter was simulated using the ARRL Radio Designer software (Compact Software sub-set) and the various dimensions were determined for popular frequencies from 1152MHz up to 2304MHz. These are detailed in tables later. The dimension P in Fig 3 will be defined later.

Please bear in mind these dimensions were obtained from computer simulations, some "tinkering" may be required to obtain the correct response for a particular filter. Details of corrective measures are contained in the sections "Tuning Up" and "Correcting Errors".

The top and bottom covers are made from aluminium sheet of 3mm thickness and contain various holes. The resonator rods are also aluminium but an unusual diameter. They are 9.525mm in diameter. 9.525mm, (which we will round to 9.5mm for simplicity), is in fact the imperial dimension of 3/8″ (0.375 in) and the reason was the need to thread the bottom ends of the resonator rods with a suitable fine thread. There is no reason you cannot use metric bar stock of 10mm but fine pitch metric taps and dies are more difficult to obtain and more expensive. Standard (coarse pitch) metric threads are too coarse and "wobbly" for precision work such as this application.

The writer has a 3/8″ x 32 UNEF set of taps and dies for threading material. 3/8″ x 32 UNEF is the thread used for BNC connectors, in fact all the popular RF connectors traditionally used are based on imperial thread types. For example, N type connectors use 5/8″ x 24 UNEF, SMA connectors use ¼″ x 36 UNS and TNC connectors use 7/16″ x 28 UNEF. The numbers 24, 28, 32 & 36 etc is the number of threads per inch. As many of these popular connectors are also specified under the NATO stock number system (NSN) they are not going to change any time in the foreseeable future!

A fact that the "Euro-crats" almost got horribly wrong recently when it was proposed to ban the use of imperial measurements or thread sizes. Fortunately common sense prevailed and the imperial thread and measurement system has been granted a new life.

Note: UNEF stands for Unified National Extra Fine, an American standard thread type, and is a finer pitch than normal UNF threads. The threads per inch for the 3/8″ UNF thread is 24. 32 threads per inch are equal to a pitch of 0.03125″ or 0.79375mm. A standard M10 thread has a pitch of 1.5mm and the Metric Fine pitch is 1mm. Some metric sizes are available in 3 different pitches. For example, M9 is available in 1.25mm, 1.00mm and 0.75mm, these being the standard, fine and extra fine versions.
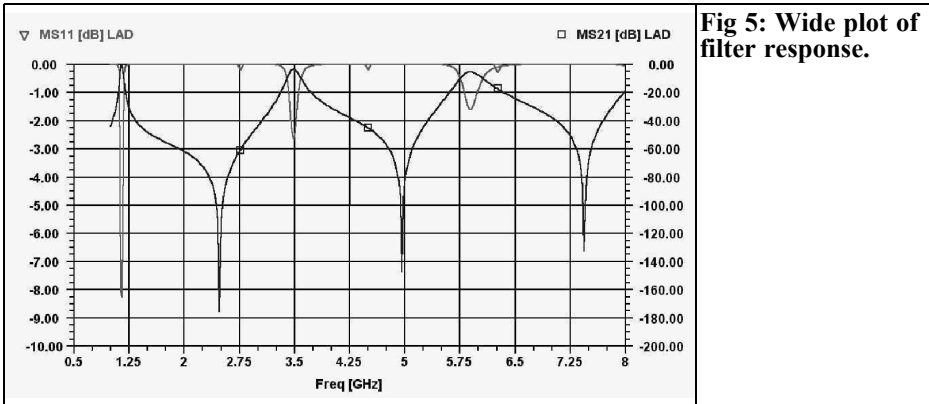
Fig 5: Wide plot of filter response.

2.0

Simulated response

The response of the 2-resonator filter for 1420MHz for a Hydrogen Line radio astronomy receiver is shown in Fig 4. You will note that the top coupling tends to make the filter behave like a high pass filter on the higher frequency part of the response plot. Normally this is of no concern as long as the attenuation obtained is adequate for the intended purpose. The ripple in the pass band can be adjusted to close to zero. In Fig 4 it is only 0.1dB maximum.

Note also the excellent impedance matching possible. The value of the input reflection coefficient (VSWR) MS_{11} is typically –30dB or better for all the filters simulated. The stop band attenuation is comparable to a 3 or 4 resonator comb line or inter digital filter but the insertion loss is much lower because more resonators equate to more loss. This means that whereas we have previously needed to use 3 or 4 resonator filters to get the required stop band attenuation we no longer have to with this new design. It also makes the filter less expensive and smaller for the same performance.

3.0

Some facts about resonator filters.

Many amateurs are not aware of the facts about resonator filters, like the comb line and inter digital filters. Many assume they are "monotonic", that is, they only respond to a narrow band centred at the operating frequency. Unfortunately this is not the case. All resonator filters have spurious responses on higher frequencies when the line appears as $3\lambda/4$, $5\lambda/4$ etc, which are not directly harmonically related, although they are close to the odd order harmonics. The reason they are not exact odd order multiples is because the fringing capacity, and any loading capacity, alter the effective electrical length as the frequency rises. Fig 5 shows a typical comb line, inter digital or this new design filter across a wide range of spectrum. This filter is one designed for 1152MHz and it has spurious responses at about 3.5GHz and 5.8GHz as well as many higher frequencies.

Hence, these types of filter are unsuitable for eliminating 3rd harmonic energy in a transmitter feeding an antenna. Also, because of the very high Q and hence impedance at the top of the resonator rods, the RF voltage may becomes exces-

Table 1: Dimensions of resonator rod length.

Frequency (MHz)	Length of resonator (mm)
1152	60.4
1268	55.9
1296	53.5
1420	48.5
1572	43.75
2160	31.8
2304	29.1

Fig 6: Details of resonator rod.

sive and may cause the filter rods to arc to ground with as little as 1W of power. So these types of filter are best used where the power levels are low. Harmonic filtering is best obtained with low pass filters such as the "Rod-Bead" filter for the µwave spectrum.

The other factor is the length of the resonator rod versus the impedance. The coaxial resonator is the easiest one to understand. The characteristic impedance, (surge impedance), of a coaxial line is determined by the physical dimensions and the dielectric material. We often use air as the dielectric (as in this case) and so the dielectric constant is 1.00. PTFE has a dielectric constant of 2.12 greater than air.

As the characteristic impedance is varied the physical length of the resonator rod also needs to be varied to maintain resonance. For a low impedance resonator the length needs to be increased to maintain resonance, conversely for a high impedance resonator the length needs to be reduced.

In coaxial resonators approaching a quarter wavelength electrical length the optimum impedance is 70.7Ω. But this is only true where the end capacity is simply that naturally existing, the so-called "fringing capacity". For a capacitively loaded, and hence shortened, quarter wave resonator this no longer holds true and we can choose any impedance within reason. In this design the impedance was chosen purely based on me-

chanical constraints. The resonator rod was chosen to be 9.5mm (3/8″) because of the availability of suitable threading tools. The simulations showed no significant deterioration as the impedance was varied over a wide range. Lower impedance made the resonator rods longer but the insertion loss in the pass band tended to be slightly less than higher impedance rods. Practicality must enter the argument eventually and the diameter of the resonator bore was chosen to be 17mm simply because a suitable drill was to hand to drill the filter block. Using the standard formula for air dielectric coaxial lines:

$$Zo = 138 \log (D/d)$$

Where:

d is the resonator rod diameter

D is the bore diameter

This gave a value of 34.71Ω with the dimensions chosen. The ratio of the two diameters is 1.78:1, as long as we stay close to this ratio then little change will be required in the dimensions of the resonator rod lengths. The software simulated the required capacitor values and the physical length of the resonator rods for the various operating frequencies. These are listed in Table 1.

The length quoted in Table 1 is the length for resonance; this is not the total length
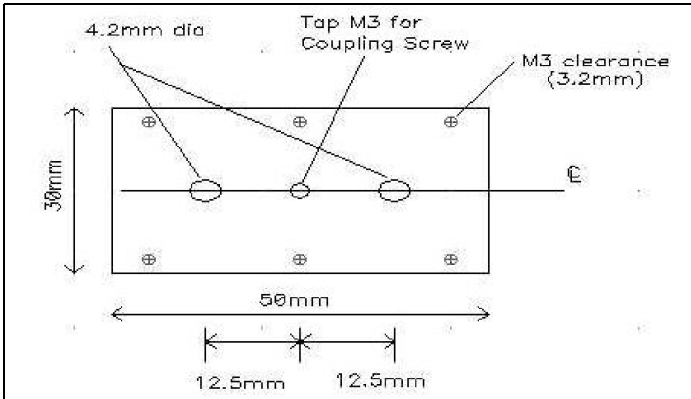
Fig 7: Top cover details. Material 3mm aluminium sheet.

of the resonator rod. This needs to be calculated from the information contained in Fig 6. The rod is threaded for a length of 16mm and the dimension from the bottom cover to the top of the rod is the dimension quoted in Table 1. This is denoted by dimension L in Fig 6.

In Fig 6 the total length of the resonator rod is the table dimension plus 8mm.

The dimension P shown in Fig 3 can now be stated. This dimension is the resonator effective length plus 6mm. For example, for 1420MHz the effective length of the rod is 48.5mm and the dimension P is hence 54.5mm, being the internal height of the cavity. This is the height of the filter block required and it should be machined to this dimension with a smooth surface finish on the top and bottom faces. An error of 0.5mm either way is not going to make the filter

not work; it simply will not work as well as it could!

Fig 7 & 8 give details of the top and bottom covers and Fig 9 gives details of the filter block.

4.0

Construction method

For this you will require certain items. At the minimum a steel ruler marked in millimeters, a marking scriber, small centre punch, small hammer, right angle square, and a selection of drills and taps. A vernier calliper if available is a good substitute for the ruler and scriber and the lines can be scribed with the tips of the
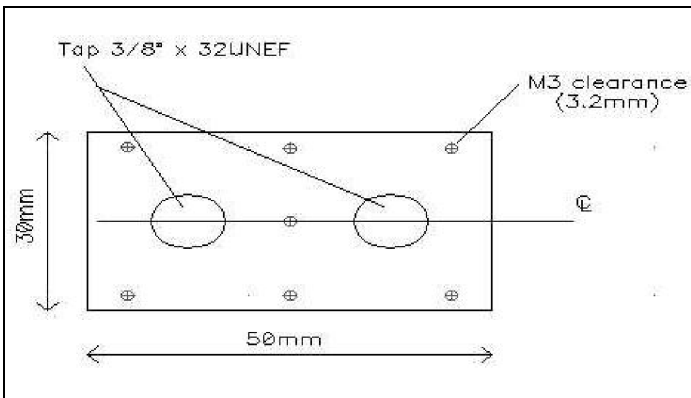
Fig 8: Bottom cover details. Material 3mm aluminium sheet.
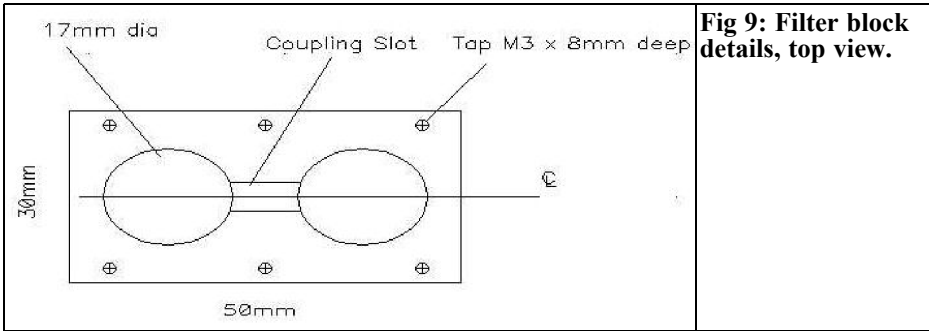
Fig 9: Filter block details, top view.

vernier jaws once set to the required dimension.

Note: Although the preferred screws are called up as metric types there are still a large number of amateurs who have British Association taps and dies and a variety of BA screws and nuts. The nearest BA size to the M3 screw is 6BA and for the M2.5 it is 8BA. BA threads are still extensively used for very small screws, especially in spectacle frames, whereas the metric sizes stop at 2mm. The smallest BA screw is 25BA, which has a diameter of approximately 0.25mm. BA screws were in fact the first available thread system based on metric dimensions and date back to the early 1850s. 0BA is the largest and is 6mm in diameter.

Fig 10: Top and bottom covers cut to size and top cover marked out and centre punched for drilling. The bores have also been marked to ensure the cover fixing holes do not intrude on the bores.

Start by cutting the top and bottom cover plates to size and then marking out the top cover. This item is then used as a jig to mark out the bottom cover and the block, so work as accurately as possible to get all the holes in the correct places. Scribe a line lengthways on the top of the cover exactly on the centre line. Find the middle of this line and then mark the centres of the two cavity bores and the coupling screw and lightly centre punch. Next mark out the 3.2mm holes for clamping the top cover to the block. The exact position of these is not too critical, just ensure you do not encroach on either of the bores! Centre punch all of these lightly and then drill all the holes with a 2.5mm pilot drill. Remove the burrs from the drilling process by rubbing the faces on a piece of 400 grit wet & dry abrasive paper resting on a smooth flat surface.

Clamp the bottom and top cover together and make witness marks with the 2.5mm drill on the bottom cover. Drill the bottom cover with 3.2mm clearance holes for the M3 fixing screws. Note that the coupling screw hole is required on the bottom cover as an additional fixing to clamp the plate between the bores.

Clamp the top cover to the undrilled block, line it up exactly and spot through all the pilot holes with the 2.5mm drill to form witness marks for the drilling.

Fig 11: Using the drilled top cover as a jig to drill the bottom cover. Note the thin piece of wood placed between the cover plates and the machine vice to prevent damage when the drill breaks through the bottom cover.
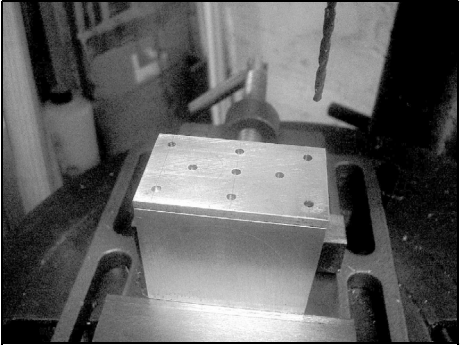
Fig 12: Using the top cover to spot through onto the block for the various holes.

Place the block to one side and continue with the top and bottom covers. It is a good idea to mark the parts in some way so you don't assemble them incorrectly. A light centre punch mark on one end of the top cover and a corresponding one on the block is one way. Whichever method you choose ensure it will not be removed in subsequent finishing operations.

Open up the 2.5mm pilot holes in the top cover to either 3.2mm or 4.2mm as appropriate. Tap the centre hole for the coupling screw M3. The tap must go through exactly perpendicular to the surface. The best way to achieve this is to

hold the tap loosely in the pillar drill chuck with the cover resting on the drilling machine table. Clamp a tap wrench onto the plain portion of the tap and use this to rotate the tap. Apply some tapping lubricant. After two or three revolutions you can finish the tapping by hand with the tap wrench fitted to the square end of the tap. Deburr both sides and give the surface that will clamp to the block a rub on some 400 grit wet & dry abrasive paper on a smooth flat surface.

Fig 13: Using a tap wrench to start the threads guided by the tap held in the drill chuck.

Fig 14: Filter block machined to size ready for drilling and two resonator rods with slotted ends and lock nuts.
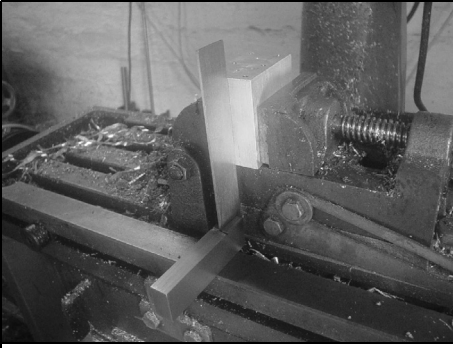
Fig 15: Checking the block is vertical in the machine vice with a set square.
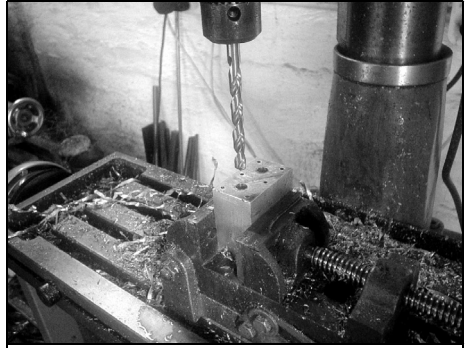
Fig 16: 6mm pilot holes drilled in block.

Treat the bottom cover in the same way and open up the 2.5mm pilot holes to either 3.2mm or 8.7mm tapping size for the 3/8″ threaded holes. Work up to the 8.7mm size in small steps so the hole is truly round and in the correct place.

Now we can drill the block. For this you will need a selection of drills varying in size up to 17mm final size. Clamp the block in a machine vice and clamp the machine vice onto the table of the pillar drill.

Do not even consider trying to hold the machine vice by hand. It will be ripped out of your hands under the cutting force and most likely cause a serious injury and damage to the block. Even if you get away without any injury the bore will not be straight and perpendicular to the top and bottom faces. Ensure the block is square and well tapped down before tightening the machine vice jaws.

Start with a small diameter drill (4.2mm) and drill as far into the block as possible. Standard length jobber drills will probably be too short for the lower frequency filters. Change to a 6mm drill and run the drill slowly, not more than 200rpm. Pour on an ample supply of cutting lubricant, I use Tapmatic #2™ for aluminium but light machine oil is also OK.

When the drill exits the bottom of the block ensure it doesn't drill into the machine vice! Position the block so the bore is in the centre of the machine vice over the hole. If you move the block during the drilling operation ensure it is tapped down square with a small hammer and piece of wood to protect the surface. A set square placed on the drilling machine table and touching the side of the block will show if the block is truly vertical.

Work up to the final hole size in small increments. We need the hole to be straight and a high surface finish. Drill out to about 16mm. When about to use the final drill, apply plenty of lubricant and run the drill as slowly as possible, for 17mm about 80 to 100rpm is about correct. Do not force the cut, let the drill work at its own pace, on the other hand do not let the drill rub, it must cut to get a good surface finish. Remove the drill bit every few millimeters of depth and clear away the swarf and apply more cutting lubricant. The better the bore finish the lower the insertion loss in the finished filter. If the bore turns out a bit ragged then you can polish it with some abrasive paper wrapped around a piece of wooden dowel. Alternatively an automotive brake wheel cylinder hone in the pillar drill will give a mirror like finish. If you have an adjustable reamer then this will also give a high surface finish to the bore. If planning to use a reamer then leave the hole slightly under-size and
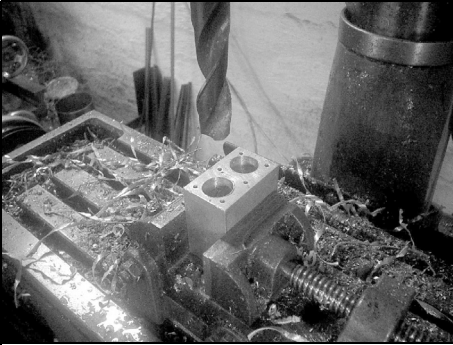
Fig 17: 17mm holes drilled in block.
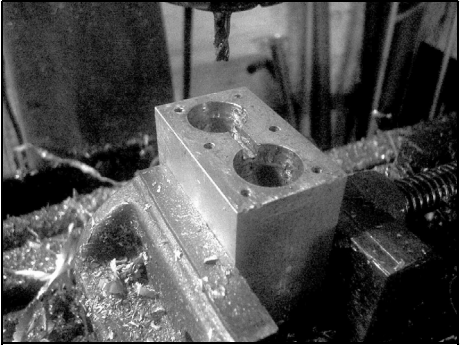
Fig 18: Coupling slot machined into top of block.

remove the last 0.2mm with the reamer to bring to finished size.

Having drilled both bores turn your attention to the fixing screws. Drill the cover fixing screws (using the witness marks made earlier) 2.5mm holes about 8mm deep and tap M3. Again use plenty of lubricant, small drills and taps break easily if used without cutting lubricant. When drilling the small holes into the block remove the drill after 2mm to clear the swarf and apply more cutting lubricant, repeat until the full depth is reached. A small paint brush dipped in cutting oil will remove the swarf and apply new cutting lubricant.

Fig 19: Threading a resonator rod in a lathe with a tail-stock die holder. The rear tool-post contains a parting off tool as a gauge for how far the threads are to be cut.

Turn the block upside down and repeat for the bottom cover holes. Remove any burring from around the bore face, a penknife is a good tool for this as aluminium is soft and easily removed.

Now the coupling iris slot needs to be machined. This needs a milling cutter 4mm in diameter. For this operation a 2 or 3 flute slot drill is the best tool. You will probably have to get this part performed by someone who has the correct tooling if you do not have access to suitable machinery. A model engineering society nearby will be the best option. Asking for assistance is not a big thing! Machine the slot 5mm deep between the two bores.

Finally rub the two block faces on the 400 grit wet & dry abrasive on a smooth flat surface to remove any burrs.

4.1 Resonator rods

The resonator rods are a simple threading operation; again a model engineering society will probably undertake this part if you do not have the correct machinery. It is important the threads are well formed and square, do not attempt this by hand! To get well fitting threads requires the threading die to be opened up slightly so the threads are slightly oversize. A lathe with a tail stock die-holder is the best method as this ensures the threads are true. Apply plenty of cutting lubricant to prevent torn threads in soft alu-

minium.

For those without access to a lathe the job can be done in the pillar drill with care. Place the die holder with the die inserted flat on the drilling machine table so it is centralised with the centre hole. Clamp the die holder down. Place the piece of rod to be threaded into the drilling machine chuck with sufficient extending for the length of thread. Bring the chuck down so the rod starts to enter the die and rotate the chuck by hand. Do not attempt this with the drill switched on! Work the drill chuck forwards and backwards to clear the swarf and apply plenty of cutting lubricant. After about 5mm of thread have been formed the rod with the die-holder attached can be removed and clamped in a bench vice and the threading completed by hand with the die holder.

Having threaded the resonator rod, the rod can be cut or parted off to the correct length and the end faced off square. If no lathe is available then a flat file placed on the drilling machine table and the rod held in the drill chuck will do almost as good a job. Ensure you do not damage the threads with the chuck jaws. Bring the rod in contact with the file with the drill running slowly.

A centre drill in the lathe tailstock drill chuck is used to make a pilot hole for the 4.2mm diameter tuning capacitor hole. Drill this 8mm deep and slightly chamfer the corner of the rod with a fine file and the entry to the hole with a centre drill. Finally, polish the rod on the plain portion to a high finish. The writer uses a piece of "Scotchbrite" pad™ made by 3M, which gives a mirror finish. The same material is found on kitchen scouring pads with the pad bonded to a piece of sponge. Apply mineral turpentine as a lubricant. Alternatively a strip of 400 grit wet and dry abrasive paper applied to the rotating rod will produce a high surface finish.

Lock nuts can be used from BNC single hole mount types, or many early potentiometers of British manufacture also used 3/8″ x 32 UNEF thin nuts and some still do. If you have access to a lathe it is a simple machining operation to make two lock nuts from 13mm hexagonal bar. If you wish you can make a saw cut across the bottom of the rods to allow a screwdriver blade to be inserted for easier adjustment. Put two hacksaw blades in parallel to form a wider slot.

4.2 Assembling the filter

Clean all the parts with alcohol or similar cleaning spirits to remove the bulk of the cutting lubricant and wash in warm soapy water (dish washing liquid) to remove any greasy finger marks. An option is to have the parts silver plated for best results; the writer hasn't had the opportunity to treat his filters this way at present.

Begin by screwing the two resonator rods into the bottom cover and fix the lock nuts loosely. Measure the distance from the bottom cover to the top of the rods and adjust to the dimension given in Table 1. Tighten the lock nuts.

Place the bottom cover on the bottom of the block and fix in place with M3 x 10mm cheese head screws, preferably stainless steel hardware. Temporarily fasten the top cover on the top of the block with two M3 screws positioned diagonally and place the two SMA connectors into the 4.2mm clearance holes for the stub nose portion. If your workmanship is OK then the centre pins should enter the hole in the rod freely. Should one or both SMA pins be out of alignment with the resonator rod, then open up the 4.2mm hole to 4.5mm and try again. If it all aligns correctly then mark the fixing holes for the SMA connectors by spotting through the connector flange with the 2.5mm drill and drill (2.1mm drill) and tap for M2.5 screws. Deburr the top cover and fix in place with M3 screws. Fasten the SMA connectors with M2.5 x 6mm screws. Ensure the ends of the screws do not protrude more than 2mm into the cavity, if too long then file them down to nearly

flush with the bottom of the top cover. Place an M3 x 12mm stainless steel screw into the coupling screw tapped hole with a lock nut. Screw the coupling screw in until it touches the bottom of the iris slot and tighten the lock nut.

5.0

Tuning Up the filter

For this we need some means of generating a low power signal that can be varied in frequency and a means of measuring the power exiting the filter. How you do it will largely depend on the type of test equipment available. A µwave signal generator and milliwatt meter such as a HP-432 is one option. The best instrument is a Vector Network Analyser, if you can get access to one for the adjustments. This will display not only insertion loss, stop band attenuation but also input and output matching for a full 2-port measurement system. Unfortunately not all amateurs have access to such luxuries!

Assuming you have the bare minimum equipment, then apply a low power signal at the centre frequency and measure the output power at the other SMA connector. Start by reducing the depth of the coupling screw. If you are lucky then the power should rise up to a maximum. Now try adjusting the resonator rod lengths by small increments. Loosen the lock nuts a little and rotate the rod either more in or more out. The power should peak some more. Measuring the power from the signal generator with the filter removed and noting the reading can make an estimate of the insertion loss. By inserting the filter the measured power will drop, by, say 1dB.

Now alter the signal generator frequency either side of the filter centre frequency. The power indicated should rise to a peak as the generator comes "on-frequency"

and drop off as we move away from the centre frequency. As we sweep across the filter centre frequency we may see a bit of ripple in the pass band as the output power varies. This is a pointer to too much top coupling as the response plot now has a big dip in the centre of the curve. The two resonator rods being slightly different in length and so resonating on different frequencies can also cause it.

5.1 Correcting Errors

So far we assumed everything was working OK. But what about the case where we have a problem? We have quite a bit of latitude to solve minor errors because we have so much which can be adjusted.

Suppose the insertion loss is excessive but the response is OK. This can be caused by several things, poor mechanical joints, which cause high resistance, and also insufficient top coupling etc. Also the input and output matching have a direct bearing on the filter ripple and insertion loss. The resonator rod as it is increased in length does two things. Firstly, it moves the resonant frequency lower, but is also increases the input/output coupling capacity as the PTFE stub is inserted further into the rod. To decrease the effect of the coupling capacity being excessive we have several choices.

Firstly, we can shorten the centre pin and PTFE stub. It is a good idea to cut the exposed centre pin, which stands proud of the PTFE, so that it is flush. This reduces the coupling a little. Secondly we can reduce the length of the PTFE stub that is able to be inserted into the rod. This is a bit drastic, because once we have cut it off we can't glue it back on again! So make a small square of aluminium 1.6 to 2mm thick the same dimensions as the SMA flange to space the SMA connector away from the top cover. Drill clearance holes for the 2.5mm screws (2.7mm) and the centre pin of 4.2mm. If this improves the

Fig 20: Completed filter.

response then we can safely trim the PTFE and pin back.

Another method is to open up the 4.2mm hole in the end of the resonator rod to decrease the coupling capacity and the rate at which it changes. Try drilling this out to 4.5mm and see the effect it has.

Insufficient top coupling, if the coupling screw is fully out, this can be corrected by machining the iris slot a little deeper or wider. It is preferable to make it wider than deeper. The reason is that if we find the coupling is now excessive we can drill out the M3 coupling screw hole on the top cover to 3.3mm and tap for a M4 screw. If the slot is too deep than a shim of a sliver of aluminium can be cut and glued into the iris slot with a drop of "Superglue" to reduce the depth. Alternatively take a small amount off the top of the block, about 1mm, and try again. Once the first filter is working satisfactorily dismantle it and note all the critical dimensions down on paper should you wish to make more filters for this frequency.

6.0

Further designs

The writer is also working on a 3-pole filter which has similar insertion loss but steeper attenuation slopes in the stop band. If sufficient interest is shown this can be the subject of another article in the future.

It may be possible to offer these filters as a commercial proposition if enough orders are placed. Cost still has to worked out but no doubt with a large enough order per annum they will be competitive if machined by CNC equipment and possibly offered through VHF Communications as a set of machined parts that just need assembling with hand tools. Contact the editor if you are interested.

John Fielding ZS5JF

John's Mechanical Gem No. 1

1.0

Introduction

Many items for the μwave bands require machinery to manufacture some of the parts. A well-equipped amateur workshop would have a small lathe and possibly a milling machine, as well as a pillar drill and a selection of drills, taps and dies. However, very few amateurs are this lucky and often need to sub-contract the parts manufacture or beg help from a better-equipped friendly amateur who has the necessary machinery.

But with a little ingenuity some tricky machining operations can be achieved with a normal pillar drill and a bit of inspiration. This is the intention of this series of short articles.

2.0

Machining resonator rod

Let us run through the operations to make a set of resonator rods (fingers) for
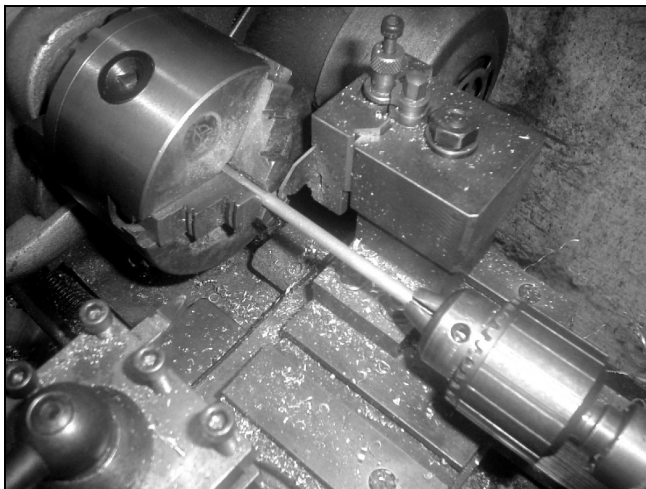
Fig 1: Parting off a resonator rod to length in a small lathe.
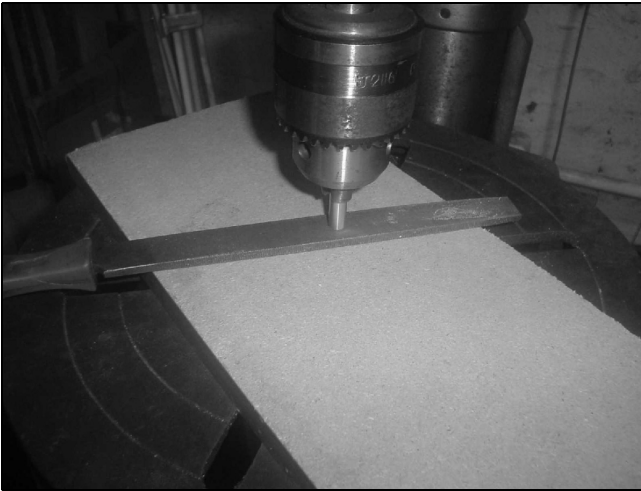
Fig 2: Using the pillar dril to square off the rod end to the correct length.

a band pass filter or similar item.

In a well-equipped workshop one would begin by cutting the resonator rods to length, with perhaps a little extra to allow for cleaning up. This is normally done using a metal cutting lathe with a parting off tool. Once the first rod has been cut to the correct length the others are a simple "repeat until enough are cut". Fig 1 shows the operation in a typical small lathe.

The rod is pulled out from the self-centering chuck until it hits an end stop; in this case the tailstock drill chuck has been locked to the length for the cut. The parting off tool is held in a rear toolpost and this is also locked so that every rod will be exactly the right length. When the first rod has been cut the lathe chuck is slackened and the remainder of the rod pulled out until it meets the drill chuck jaws, the chuck is tightened and the next rod can be parted off. With a setup like this the error is less than 0.05mm between rods.
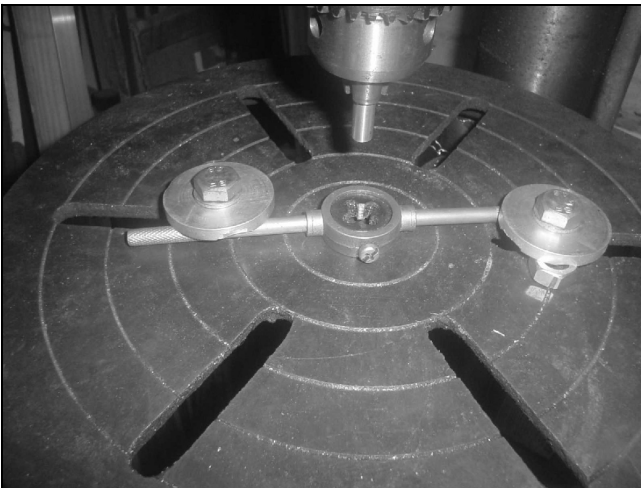
Fig 3: Die holder clamped to the drill table.
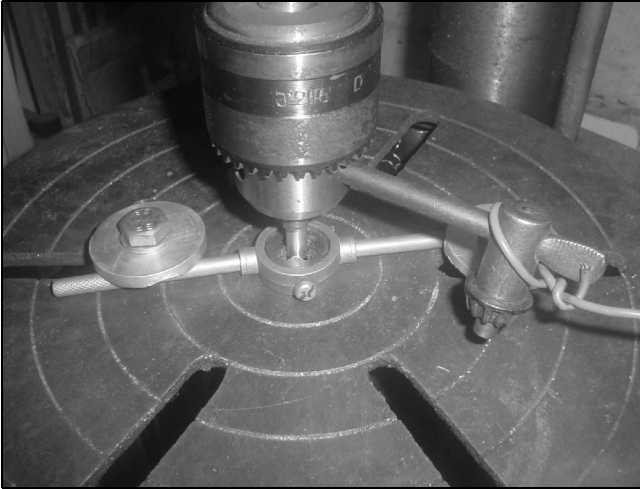
Fig 4: Using the drill chuck key as a lever to rotate the chuck.

The second step may be threading one end to screw into the filter block. In a lathe this utilises a die holder held in the tailstock. This ensures the threads are truly square to the rod and not "all wobbly". If the threads are not square to the rod the resonator fingers will lean over at a drunken angle, so this is important.

3.0

Using simple tools

When a lathe is not available we can do as good a job with a simple pillar drill. Many amateurs seem not to be able to saw straight, so cutting off the rods by hand with a hacksaw leaves the cut end not square to the rod. This can easily corrected with the setup shown in Fig 2. The rods should be cut slightly longer than the finished length and the exact length marked on the rod with a scriber or sharp pencil.

A piece of wood is placed on the pillar drill table and a fine flat file placed on

the wood. The drill is run at medium speed and the rod brought into contact with the file. By applying a little downward pressure the rod is "machined" exactly square and to the correct length. Before removing the rod place the file at an angle of 45° to remove the burring caused and to leave a small chamfer. In practice the operator would hold the file firmly on the wood to prevent slipping. This is a posed shot and I only have one pair of hands!

Now we have to cut a thread on one end. Many amateurs are not capable of producing a truly true thread using a die-holder by hand. So a little extra setup allows a job as good as the lathe.

Fig 3 shows the die-holder placed on the pillar drill table and clamped down. The concentric circles machined in the table give a good indication of the die-holder being correctly concentric with the center hole in the table. To assist in aligning the die-holder, place the rod to be threaded into the drill chuck with about 25mm sticking out. Now with the drill chuck lowered it should go into the die exactly in the middle. If not slacken the clamping bolts and move the die-holder until it does.

Note that in Fig 3 a nut is placed under each washer to prevent it "cocking over" and slipping. The washers are off cuts of 35mm aluminum rod 6mm thick with a hole drilled in the center to suit the clamping bolts. Normal black-iron fender washers could also be used if two or three are stacked to increase the thickness.

Fig 4 shows the threading about to commence. The drill chuck key is being used as a lever or Tommy Bar to rotate the drill chuck by hand. **Note: Do not attempt to thread the rod using the drill powered up!**

A suitable cutting lubricant should be applied and the drill chuck worked forward and backward to break the swarf. When about 5mm have been threaded the die-holder with the rod still attached can be unclamped and the job moved to the bench vice to finish cutting the threads by hand to the required length. Because the die was started truly square to the rod it will continue along the same path for as long as required.

Gunthard Kraus, DG8GB

Internet Treasure Trove

TutorialsWeb

The name says everything: tutorials are collected here and the topics are so varied that a thorough search is worthwhile. You will find some very useful topics covered that will be extremely useful. The scope of the topics will not be revealed here by suffice to say that the web site is well worth visiting.

Address:
http://www.tutorialsweb.com/index.html

Noise and Noise Figure Measurement

A very good detailed homepage. This is nearly a complete textbook online. Great fun to read either for the first time or to refresh knowledge from the past.

Address:
http://www.rose-hulman.edu/~black/Noise/ppframe.htm

Propagation Tutorial

A very well presented and easy to read tutorial with plenty of information on antennas, wave equations, propagations, reflections, Noise….?

Address:
http://www.mike-willis.com/Tutorial/PF13.htm

BesserNet

You will find many application notes here suitable for the experienced developer or the experimenter. Pleas sepne some quiet time to go through the appropriate pdf documents or download them to read later.

Address:
http://www.bessernet.com/jobAids/resourcesAlpha.asp?Type=1

Agilent

No introduction is needed to this company formerly known to us as Hewlett Packard. The company is still growing and to emphasise that there are (on 1 September 2007) exactly 2946 application notes available to download on this homepage. For those who are interested of do not believe this, visit:

Address:
http://www.home.agilent.com/agilent/facet.jspx?lc=eng&cc=US&t=80030.k.1

Designing Antenna systems with maximum sensitivity

This is a very well presented and informative homepage. There are many interesting subjects mentioned on the homepage.

Address:
http://www.pentodepress.com/receiving/receiving.html

COMM Design - Design of center

This is a collection of useful articles and training material from different companies. You will find lots of useful information here.

Address:
http://www.commsdesign.com/dcenters/wireless/

Cisco - Carrier to Noise ratio in Cable network

Companies like Cisco who have their roots in computer and network technology make good tutorials on RF and communications available. A good example is this analysis signal to noise ratio. There is more information on this web site, so explore the home page.

Address:
http://www.cisco.com/en/US/products/hw/cable/ps2211/products_white_paper0900aecd800fc94c.shtml#wp9000208

Fast pdf reader

An alternative to the well-known pdf reader is the very small (only 2.1Mb download) and quick software FOXIT reader from the company FOXIT software.

Apart from the free Windows version there is a versions for mobile devices, desktop Linux and U3.

This is worth a look:

Address:
http://www.foxitsoftware.com/pdf/rd_intro.php

Satori Technology high dynamic range USB 26.5GHz power meters

Field pioneers Satori Technology have extended their product range of USB

power meters with the release of the ST265. This pushes the frequency limit for high dynamic range USB power meters up to 26.5GHz going beyond what is currently available in the marketplace. The ST265 now covers a frequency range of 10 MHz to 26.5 GHz.

The ST series are complete miniature RF & microwave power meters. All measurement data and settings are sent via a USB interface to and from a PC or laptop. This gives the concept of using a computer as the user interface in classic microwave power measurement. Each power meters contains a CPU which controls the sensor, processes the measurement results and operates the USB interface.

The ST265 is a significant enhancement to the existing series (ST124 and ST185 USB Power Meters) and brings with it greatly improved performance. Extended individual calibration procedures have resulted in improved temperature compensation and better linearity performance with RF frequency. The use of a K-typeTM RF connector gives repeatedly low reflection coefficients up to 26.5GHz.

Satori Technology supplies its own easy to use software toolkit which is powerful, flexible and enables multiple power meters to be operated simultaneously. Drivers are available for Agilent VEE™ and National Instruments LabVIEW™. With a size of only 34 x 43 x 125mm and weighing in at only 83g the ST range of power meters are the ultimate in portability as well as saving space and reducing costs in rack-mounted systems.

Address:
www.satori-
technology.com/products.htm

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!

The Satori ST265 USB power meter
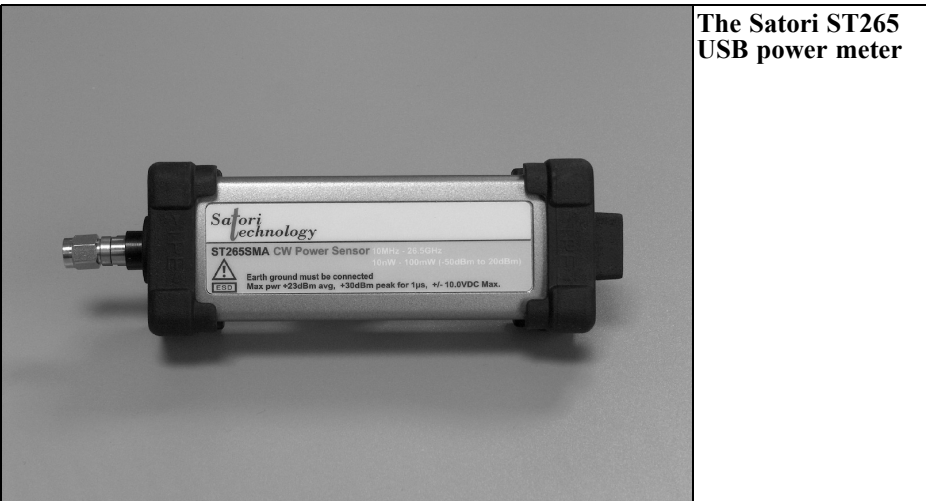
VHF COMMUNICATIONS

A Publication for the Radio Amateur Worldwide

Especially Covering VHF, UHF and Microwaves

Volume No.40   Spring   Edition 2008-Q1

ISSN 0177-7505

Back Issues

All issues ever published are now available either as photocopies or actual magazines. Issues from 1/1969 to 4/2006 are £1.00 each + postage. Issues from 1/2007 to 4/2007 are £4.70 each or £18.60 for all 4 issues + postage. See web site or page 34 of issue 1/2008 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 £1.55, Surface mail £1.85, Air mail to Europe £2.20, Air mail outside Europe £3.30)

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 or CD with a full English handbook. PUFF is £20.00 + postage. (UK £0.60, Surface mail £1.30, Air mail to Europe £1.35, Air mail outside Europe £2.20)

Compilation CDs

Two CDs containing compilations of VHF Communications magazine articles are available. CD-1 contains 21 articles on measuring techniques published over the last 8 years. CD-2 contains 32 articles on transmitters, receivers, amplifiers and ancillaries published over the last 5 years. The articles are in pdf format.

Each 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.

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.

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

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