

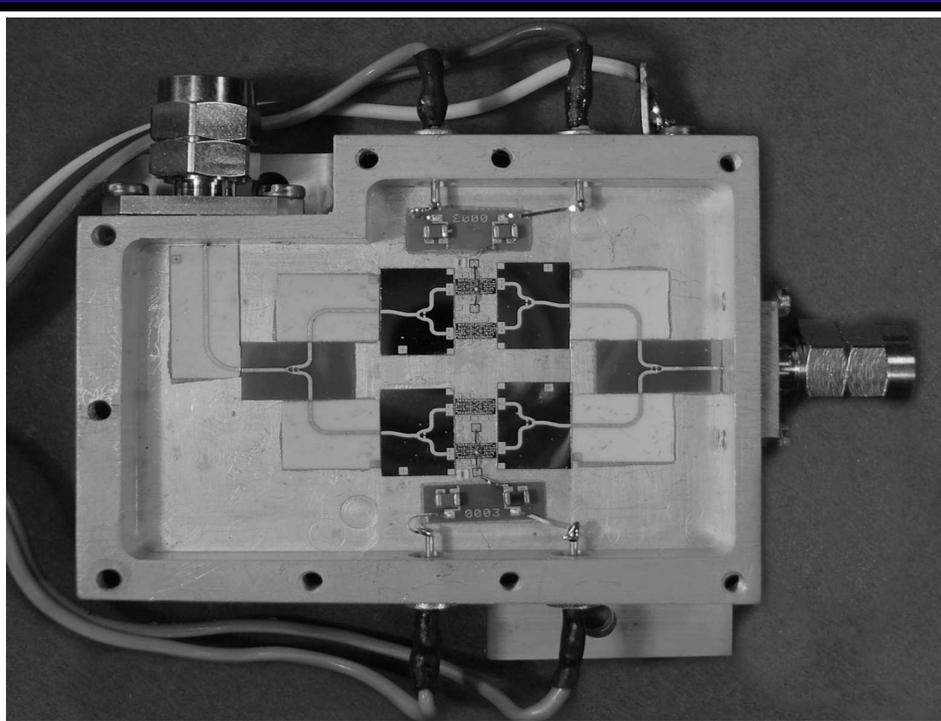


*A Publication for the
Radio Amateur Worldwide*

*Especially Covering VHF,
UHF and Microwaves*

VHF COMMUNICATIONS

Volume No.45 . Autumn . 2013-Q3 . £5.70



**Oscillator for 122GHz: Frequency multiplier from
61GHz and amplifier**

Sigurd Werner, DL9MFV

Practical WIRELESS

Britain's Best Selling Amateur Radio Magazine

Enjoy the World of VHF every month!

Are you missing out on **Tim Kirby G4VXE's** *World of VHF*? You are if you're not reading *Practical Wireless* regularly!

Rob Mannion G3XFD, Editor of *PW* invites all v.h.f. enthusiasts to join our keen contributor. "Tim G4VXE has blown an exciting breath of new life in to his column as he explores the fascinating spectrum above 30MHz. It literally fizzes with enthusiasm and dedication and – as an extremely keen v.h.f. operator, enjoying the wide number of activities on the v.h.f., u.h.f. and microwave bands – Tim G4VXE writes a fully inclusive column. There's **something for everyone** in the *World of VHF*, whether you operate on f.m., s.s.b. a.m. or Morse and data modes. He proves – every month – how Radio Amateurs using relatively simple v.h.f. installations can work the DX!

However, as Tim clearly reports – there's much more to the *World of VHF* than chasing DX. Everything – from mobile and portable working to propagation reports – are enjoyed by Tim's growing band of readers around the globe. Tim's readers are a central part of the *World of VHF* team. So, join in and make sure you don't miss out!"

Subscribe today!

- Never miss an issue
- Have it delivered to your door
- Subscribers get their copies before they reach the shops
- Avoid price rises
- Save £££s

To order a subscription please contact our subscription agency:

Practical Wireless Subscriptions
Unit 8 The Old Silk Mill
Brook Street, Tring
Hertfordshire HP23 5EF

Please note: any cheques should be made payable to PW PUBLISHING LTD and CASH is NOT accepted by ourselves or Webscribe.

Orders taken on:

01442 820580

between 9am – 5pm. Outside these hours your order will be recorded on an answering machine.

FAX orders taken on:

01442 827912

Secure Internet orders can be placed at:

www.mysubcare.com

or via e-mail to:

pw@webscribe.co.uk

PLUS REVIEWS, FEATURES AND ALL THE REGULAR FAVOURITES

Visit www.pwpublishing.ltd.uk
for up-to-date issue contents and more information



Contents

Sigurd Werner DL9MFV	Oscillator for 122GHz: Frequency multiplier from 61GHz and amplifier	130 - 135
Gunthard Kraus DG8GB	An Interesting Program: Today PUFF 2.1 for Windows 7; now on CD	136 - 147
Aristotles Tsiamitros	Operational Transconductance Amplifiers (OTA)	148 - 162
Dirk Müller DB6FM	The RTL-SDR Working with a USB stick Part 2 continued from issue 2/2013	163 -173
Harald Braubach DL1GBH	Directional couplers - made to measure Reprint from 4/1984	174 - 180
Michael Martin DJ7VY	Extremely Low Noise Preamplifiers require Low Loss Antenna Cables Wideband Directional Coupler for VSWR Measurements on Receiver Systems Reprint from 3/1983	181 - 189
Gunthard Kraus DG8GB	Internet Treasure Trove	190 - 191

There have been no more entries in the article writing competition so there are just 3 for you to vote for either by post or on the web site.

Because there were no more entries I have reprinted 2 articles from past issues.

I have only had one person interested in taking over this publication at the end of the year. The magazine will therefore be merging with DUBUS magazine. If you want to continue to read articles from UKW Berichte you can subscribe to DUBUS magazine. There will be more details in issue 4/2013.

73s - Andy



K M Publications, 503 Northdown Road, Margate, Kent, CT9 3HD, UK

Telephone / Fax +44 (0)1843 220080, email : andy@vhfcomm.co.uk

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



Sigurd Werner, DL9MFV

Oscillator for 122GHz: Frequency multiplier from 61GHz and amplifier

The design of a passive frequency doubler and a sub-harmonic mixer for 122GHz requires a strong signal on 61GHz. Two multipliers using several amplifiers were based on different frequencies (10.2 and 15.3GHz) and built for the planned 122GHz doubler. Different strategies are presented for 61GHz oscillator processing.

1.

Mechanical and electrical construction

The housing for the modules was milled brass and plated in an alkaline medium (pH 8.2, at 72 degrees).

The substrates used were Al_2O_3 , AlN and Quartz (0.254 or 0.127mm). The Al_2O_3 and AlN ceramics were cut for me on a guided XY table (Diamond Ritz device, ATV, technology). The quartz substrates were cut to the correct size from the wafers (see section 6). The mechanical level between MMICs and the ceramics was adjusted by reducing the height of the substrate. MMICs and ceramics were attached with silver conductive adhesive (Ablebond 84-1LM3) at the appropriate points of the modules. Chips and conductors were connected by bonding (Highbond, model 572) using 13 μm gold thread. Impedance adjustments were carried out by applying small “gold flags” on the strip lines as necessary and fixed with

silver conductive adhesive. The DC power circuits suggested in the data sheets for the MMICs were used (single layer ceramic capacitors and gate resistors) and coupling between modules with capacitors. Detailed descriptions can be found in earlier publications [1-4].

2.

Measurements

Performance measurements were performed with an MP716A V band sensor and an ML4803A power meter by Anritsu. For coaxial connection a waveguide to coaxial transition from Suncor was used and switches (MI-wave, 410V, about 0.7dB attenuation). A variable attenuator by Hughes was used to measure strong signals. The spectral properties of the RF output was measured with an HP spectrum analyser (70000 system with harmonic 11970V mixers).

3.

Signal sources for 61GHz

3.1. Module A

The module was designed as a six times multiplier (Fig 1). The input frequency is approximately 10.2GHz (4 - 5dBm). The signal is tripled by a CHX2093a MMIC

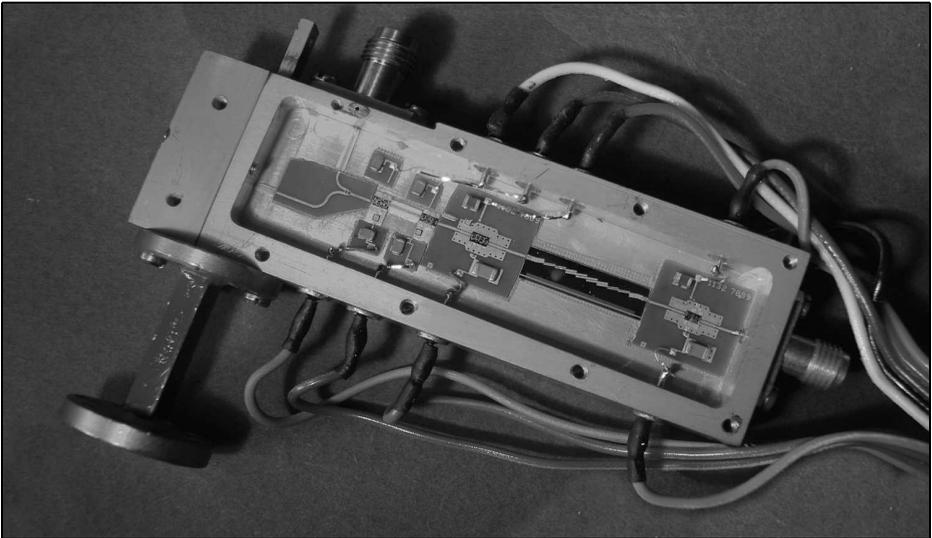


Fig 1: Module A. A six times frequency multiplier from 10.2 to 61GHz. The input signal is fed via an SMA socket (bottom right). The frequency is trippled, the signal is filtered (30GHz stripline filter), amplified, doubled to 61GHz and amplified. The MMICs used are clearly visible. A Wilkinson divider splits the output signal between a 1.85mm coaxial connector and A V band waveguide.

(United monolithic semiconductors, UMS), followed by a quartz strip line filter for 30GHz (insertion loss approximately 3.2dB). The power is increased to approximately 12dBm by one chip (CH-X2092a - UMS, or HMC518 - Hittite) and then doubled to 61GHz by a third MMIC (CHX2192 - UMS). This signal is increased by an additional chip (HMC-ABH209-Velocium/Hittite) to approximately 16 - 17dBm and then fed into a Wilkinson divider. The split power feeds a coaxial socket (1.85mm V socket, Anritsu) and a waveguide (V band). The output was measured as 12 - 13dBm from both outputs.

3.2. Module B

This module was designed as a four times multiplier (Fig 2). It is fed via an SMA female (left of Fig 2). The input frequency is approximately 15.3GHz. The MMIC HMC XDH158 (Hittite) quadruples this to 61.2GHz. The output signal (approximately -3 to -4dBm) is filtered using a 60GHz quartz strip line

filter (with only 4.1dB attenuation!). This is followed by an amplifier chip (HMC ABH241 by Velocium / Hittite, 22 dB gain) bringing the output to 13 - 14dBm. Due to this high RF gain, in combination with the geometry of the structure, there is a significant tendency for the module to oscillate. This problem can be fixed by two measures. The amplifier chip was screened from the MMIC by a partition in the housing and an upstream 3dB attenuator (stripline "T" construction). Thus, a stable output was achieved of approximately 13dBm at 61GHz. The signal output is a 1.85mm connector (right of Fig 2).

4.

Power amplifier for 61GHz

The first attempt used a Wilkinson divider feeding two amplifier chips (HMC-

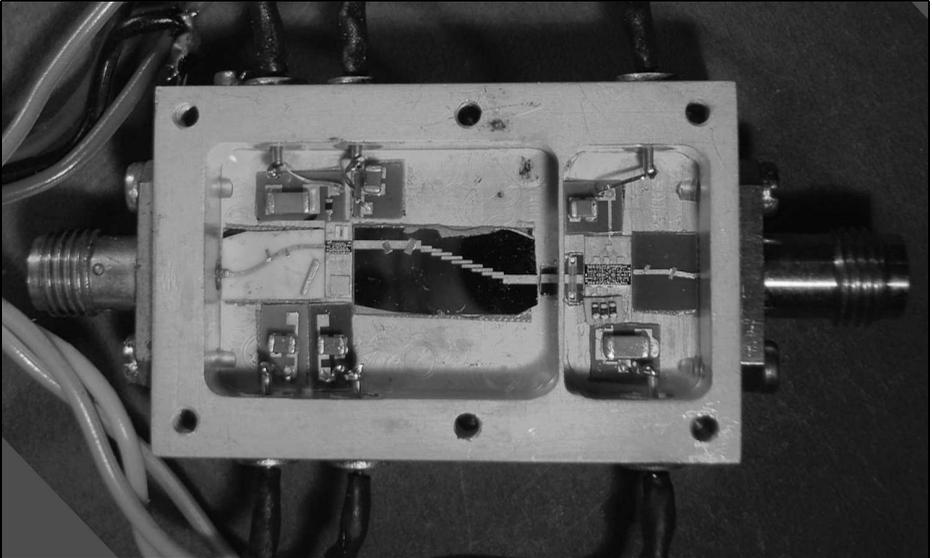


Fig 2: Module B: Frequency quadrupler, the stripline filter, in a small housing, the 3dB attenuator and finally the amplifier MMIC.

ABH209) in parallel separately coupled to the V band rectangular waveguide. An output power from 18.5 to 19dBm encouraged me to build module C shown in Fig 3.

4.1. Module C

The 61GHz signal is fed in via a 1.85mm V plug (right of Fig 3) and amplified to approximately 16dBm with a driver HMC-ABH209. This feeds four parallel

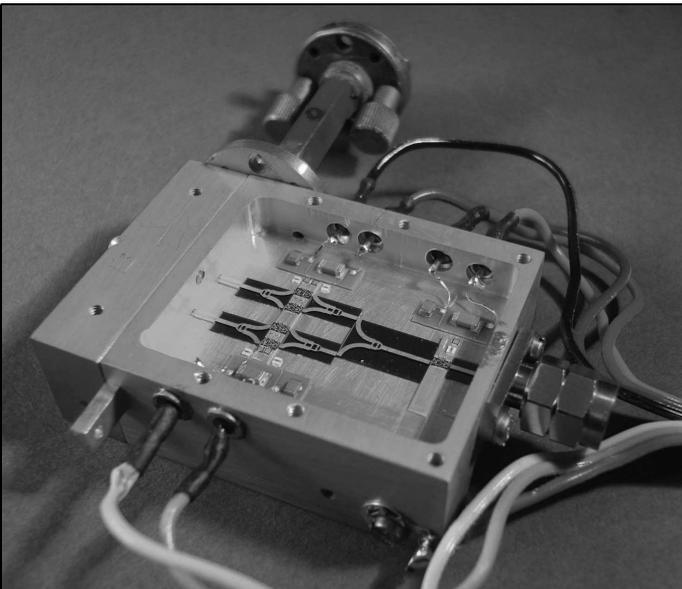


Fig 3: Module C: Power amplifier for 61GHz. The signal is fed in at the right to a driver chip then to four parallel MMICs and then to the waveguide. The two blocks forming the side of the waveguide can be seen. The slightly raised platforms giving a level surface between the MMICs and the quartz substrate for the chips can be seen.

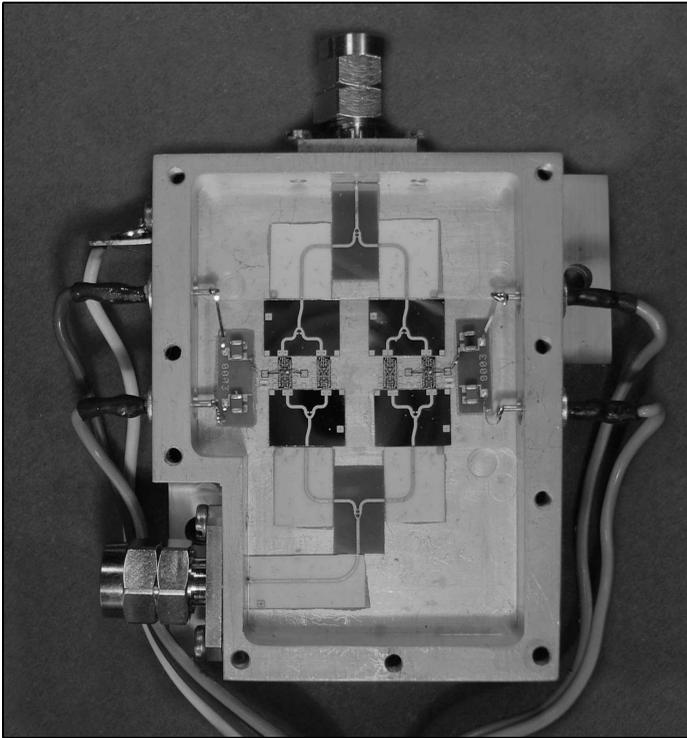


Fig 4: Module D: Power amplifier for 61GHz. The signals input and output is by coaxial connections. Wilkinson dividers distribute the power.

power amplifier chips (HMC-ABH209) via Wilkinson dividers. The output of two pairs of MMICs are merged and then coupled into the waveguide spaced at a distance of $1\lambda_g$ apart (left of Fig 3). Despite looking like substrates pieced together the output was 20.9dBm (120mW). This is more than the Schottky diodes to be used to double need (they run into "saturation" at 19.5dBm).

4.2. Module D

This design (Fig 4) has two changes from module C: the input and output are coaxial; the driver stage could be removed because the more powerful HMC-ABH241 (Velocium/Hittite) MMICs were used for improving performance. The four Wilkinson dividers, designed for 76GHz, were on the MMIC substrates (0.127mm quartz). They work extremely well between 60 and 90GHz. However, the four high gain chips in the chosen environment tend to oscillate badly (des-

pite the use of ample damping material), so they could only be operated at reduced power. The power output achieved was disappointingly low.

4.3. Module E

This was a "scaled down" version of module D using only two HMC-ABH241 chips (Fig 5). The coaxial module is extremely compact (28mm x 25mm) and requires only 6 to 8dBm input power. The light oscillation behaviour of this module can be easily mastered. The signal at the output is more than 120mW.

4.4. Module F

This design uses some unusually designed substrates for pairing of the four MMICs. The relatively narrow band (0.1-27mm quartz substrate) structures were originally calculated for 70GHz. However, since the RF simulation (Sonnet) at a frequency of 60 to 61GHz predicted

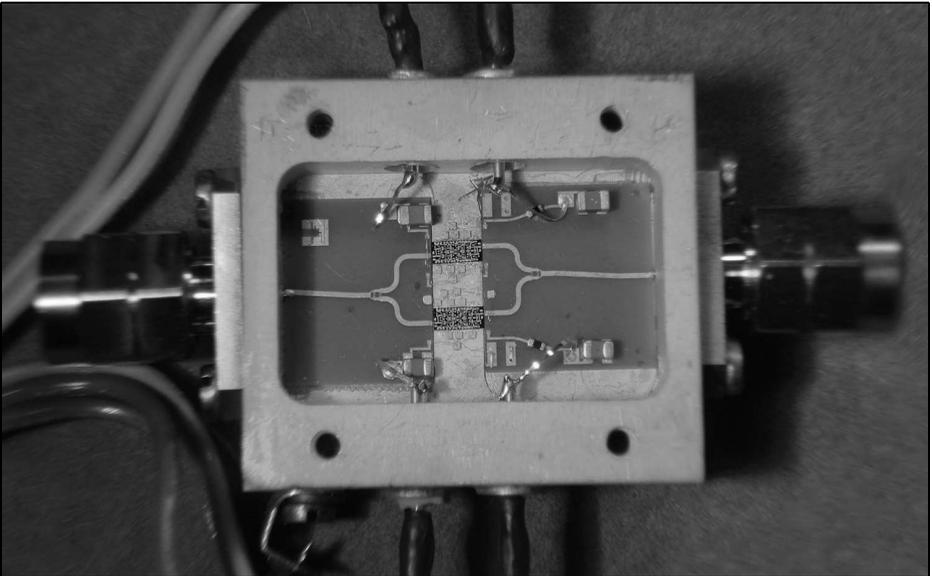


Fig 5: Module E: Power amplifier for 61GHz. This compact coaxial model is fed by the 1.85 mm connector and requires only low driver power.

relatively low loss properties, this module was built (Fig 6). Driver IC and amplifier MMICs are as described in module C. Almost 120mW was measured at the coaxial output of this module, which is the same value as in module C. A test with other MMICs at 70GHz is on hold.

5. Generation of the output frequencies of 10.2 and 15.3GHz

This is a rather trivial issue for those familiar with microwave construction, just two options are used in practice.

The direct use of a frequency synthesiser coupled to a 10MHz reference is easiest for 10.2GHz.

In this case the author used 15.3GHz from a PLL stabilised fixed frequency oscillator. The signal at 119,244MHz

was amplified, doubled and filtered. The power at 238,488MHz was 8dBm and fed to a phase controlled oscillator (HMC535LP4, Hittite) that produces 64 times the input frequency giving approximately 10dBm at 15.263GHz. Dual oven crystals are good for the stable 10MHz reference signal but connection to a GPS signal is also a popular solution.

6. Thanks

I would like to thank the employees of Rohde and Schwarz, Munich, especially Dr. Hechtfisher (DG4MGR) and W. Hohenester for supplying the ceramic substrates and the 5 mil quartz for the various power couplers. W. Hohenester generously carried out the sawing of the sensitive quartz substrates. The PLL stabilised fixed frequency oscillator for 119.-244MHz was made by Michael Kuhne (DB6NT). Finally, I wish to thank my

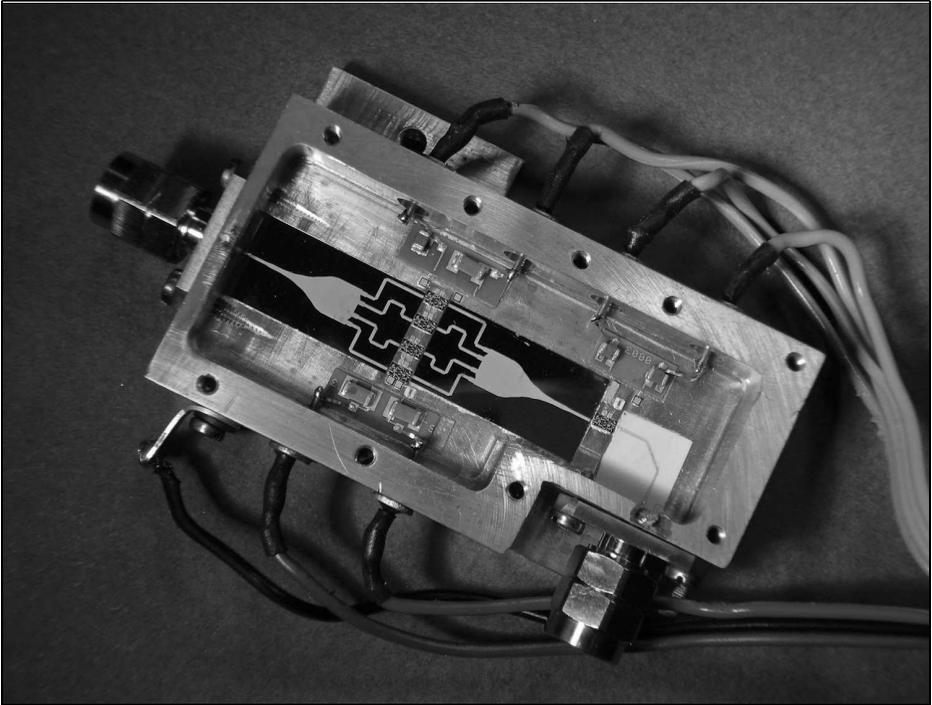


Fig 6: Module F: Power amplifier for 61GHz. This experimental setup does not use Wilkinson couplers but a special multiport system. The RF is fed to the coaxial connector and then to a driver MMIC and then the four amplifier chips. Energy losses in these quartz substrate are similar to those of the Wilkinson system used in module C.

friends Max Münich (DJ1CR, Max Planck Institute for Physics, Munich) and Jochen Ehrlich (DF3CK) for their ongoing support and advice.

7.

References

[1] New transmitter with higher performance for 76GHz, Sigurd Werner, DL9MFV, VHF Communications Magazine 4/2003, pp 194 - 198

[2] Combining power at 76GHz: Three possible solutions discussed, Sigurd Werner, DL9MFV, VHF Communicati-

ons Magazine 1/2004, pp 13 - 19

[3] Frequency doubler for 76GHz with 130/160mW output, Sigurd Werner, DL9MFV, VHF Communications Magazine 3/2004, pp 150 - 155

[4] 76GHz sextupler and amplification, Sigurd Werner, DL9MFV, VHF Communications Magazine 3/2008, pp 155 - 159



Gunthard Kraus, DG8GB

An Interesting Program: Today PUFF 2.1 for Windows 7; now on CD

The resurrection of old DOS programs under Windows 7 was described in the last issue of the VHF Communications Magazine. After a short time using the well-known PUFF software caused me to worry about the new appearance. Then I had the idea of offering PUFF together with detailed instructions and other accessories on a user friendly CD.

1.

Introduction

PUFF is a RF and microwave CAD program that was developed for DOS and is still interesting even under Windows 7. It works in the "frequency domain" and calculates the S parameters of a circuit. It also contains: a microstrip calculator, a simple sweep facility for optimisation, the possibility to calculate step and impulse response and the output of S parameters as series or parallel equivalent circuits. Due to its simple and fast operation, but also due to the high accuracy of its calculations it was often classified as an "RF Calculator". It now comes in the form of a CD including the required installation files, installation instructions and in addition many important and useful things such as:

- An additional program to represent a

parameter as a "Bode diagram" (bode plot; separate representation of amplitude and phase)

- A large private collection of S parameter files from the Internet
- The free graphics program "Irfan view" to print out the screen and the results
- Filter and antenna calculator software. Also the data sheets and S parameters files of the ICs used in the tutorial are added
- A compilation of useful device files (such as: current source, voltage source, voltmeter, differential amplifier, emitter follower, etc.) that can be easily incorporated into PUFF, if they are not already integrated
- A collection of the authors publications (articles that use PUFF) from VHF Communications Magazine
- A PDF version of the original English PUFF manual
- A newly compiled alphabetic keyword index of the main control commands under Windows 7
- A newly written tutorial with over 100 pages and many projects. Let's not talk of the price. The CD is available either online from my web site [1] or from VHF Communications Magazine [2].



What can't PUFF do?

- No result charts with a logarithmic frequency axis
- No noise simulation and stability calculation
- No support of the image editing of a Windows program (using paste, copy...etc.). But a fine and easy method of editing a screen shot with a graphics program is included on the CD as "Irfan View"!
- No mouse... everything done with keyboard shortcuts and arrow keys
- No automatic adjustment of "Open end extensions" for micro strip lines or the use of T branches or the automatic "mitering" of bent lines. But the original manual contains easy-to-use charts for the necessary design corrections with examples in the tutorial.
- No vast quantities of component models and options like most modern microwave CAD programs (e.g., ANSOFT Designer or ADS, etc.).
- No sophisticated optimising options, only a simple sweep facility to vary a parameter

Also the file organisation when saving or opening files leaves a bit to be desired, but this was just under DOS.

Please note: how in ancient times file names with only up to 8 characters and no special characters are allowed when saving your own design.

By the way: The tutorial, software and other files can be downloaded individually directly from [1]. This is especially for those who still have the PUFF disc somewhere in a drawer and want to install everything themselves. For everyone else, the new CD is the better and more convenient solution. The special addition of approximately 450Mb of S parameter files that have been accumulated over the course of the years are only included on the CD. Another thing: Not only German, but also an identical Engl-

ish Edition including an English version of the tutorial are on the CD.

2.

Concept of the "PUFF21 for Windows 7" CD

These three folders appear when the CD is started:

- German
- English
- S Parameters and Datasheets

The German and the English folders have the following sub folders:

- Installation
- PUFF tutorial
- Info more software

Thus, the way forward is clear; please read the following in the installation folder first:

- All information about the installation
- All files for the PUFF software

In contrast to modern pure Windows programs, DOS box (for running DOS software) and a DOS shell (DOS interface) must be installed. Make a PUFF folder on the hard drive and copy everything from the PUFF file folder of the CD and then link this to the DOS box and the DOS shell. That's good and it is really as easy as it sounds and it runs perfectly.

By the way: "PUFF21" means "PUFF version 2.1". It is the last completely redesign and conversion to "protected mode - operation" version with "Bugs" removed and improved appearance of the charts. The most important change was the extension from 500 to a maximum of 1000 simulation points.

**3.****The new PUFF tutorial**

That was probably the biggest task because a cross section of the most important

practical applications in RF and microwave technology should be offered. Because again and again something occurs to me, you should regularly check the PUFF folder under [1] to see if a new version is available. The good part is the table of contents that shows what is to come: (each chapter and page number)

Content	Page
1. Introduction	4
2. Installation	5
3. First Example (without PCB): design of a Lowpass Filter	8
3.1. Filter design using the DOS program „fds.exe“	8
3.2. Simulation of the ideal LPF using PUFF	10
3.3. Simulation of a lossy LPF ($Q_L = 75$)	14
4. Getting started with the Simulation of Circuits on PCBs	16
4.1. Some Substrate Information	16
4.2. How can I get a perfect 50Ω Microstrip Line?	17
4.3. Matching of the Radiation Resistance for a GPS Antenna using a perfect $\lambda/4$ Line	21
4.4. Matching of the Radiation Resistance for a GPS Antenna using a realistic $\lambda/4$ Line	23
5. Twoports and PUFF 25	
5.1. S-ParameterFile (*.s2p) and scattering parameters	25
5.2. An Amplifier with an INA03184 at 1575MHz	27
5.2.1. Using S-Parameters in PUFF	27
5.2.2. How to improve S22	28
5.2.3. How to improve S11	30
5.2.4. Important Information for creating a PCB up to 10 GHz	31
5.2.5. A well known Problem: self oscillating of an MMIC Amplifier	33
5.2.6. The Circuit Diagram and the PCB for this Amplifier	35
5.2.7. The Truth: a Prototype and a Comparison of simulated and measured S Parameters	37
5.3. Device File, S2P File and Puff File: Differences and Conversions	39
5.4. Amplifier Design using another MMIC (MGA86576) including noise matching	40
5.4.1. Stability	40
5.4.2. Improving the Output Reflection S22	42
5.4.3. Noise matching at the Amplifier Input	43
5.4.4. Open End Extensions	48
5.4.5. The complete Circuit Diagram	51
5.4.6. The PCB	52
5.4.7. An alternative Matching Solution: Parallel Connection at the input	54



6. Design of a WLAN Patch Antenna as Example for a Oneport	56
6.1. At first: Antenna Fundamentals	56
6.2. Fundamentals of Patch Antennas	57
6.3. Measuring of Antenna Properties	59
6.4. Design of a WLAN Patch Antenna for linear Polarization	60
6.4.1. Patch Design using PUFF	60
6.4.2. Matching to 50Ω	64
6.4.2.1. Matching by a Quarter Wave Line	64
6.4.2.2. Coaxial Feed from the lower Side Mathematical Solution	67
6.4.2.3. Coaxial Feed from the lower Side PUFF Solution	70
6.4.3. How to determine the Power-Bandwidth	72
6.5 Design of a Patch Antenna with more Width and more Bandwidth	74
7. The Wilkinson Divider	78
7.1. Principles and Task	78
7.2. Example: a Wilkinson Divider for $f = 2450\text{MHz}$	78
7.3. Layout Problems	80
8. An „Edge Coupled Microstrip Bandpass Filter“	83
8.1. What is that?	83
8.2. Testing the example file „BPF_MSTR.PUF“ on the CD	83
8.3. Like a Flash of Lightning: Conversion of the Centre Frequency to 2450MHz	87
8.4. A frightening affair: using FR4 as PCB	88
9. Design of a Microstrip Low Pass Filter with a Cutoff Frequency of 1750MHz	90
9.1. At first: Filter Design using a Filter Calculator	90
9.2. Line pieces replace discrete parts	91
9.2.1. Creating “Inductors” as Line Pieces using PUFF	91
9.2.2. Creating “Capacitors” as Line Pieces using PUFF	93
9.3. First Test of the complete Circuit	95
9.4. Preparing the Layout	101
10. RF Oscillator Simulation using PUFF	103
10.1. Fundamentals of Sine Wave Oscillators	103
10.2. Oscillator Design Principles mostly used	103
10.3. Oscillator Design using PUFF	106
10.4. Analysis of a given Oscillator Circuit	107
Appendix 1	
Catalogue of Keystrokes and Commands for PUFF using Windows 7	112
Appendix 2:	
Overview of included Device Files in PUFF	121

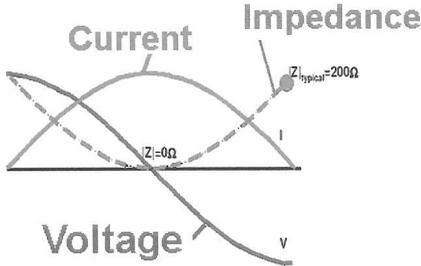
The following extract from Chapter 6 (development of Wi-Fi patch antenna) shows the basic handling of the program

and demonstrates the style of the new tutorial. Have fun with it!



6.4.2.2. Coaxial Feed at the Lower Side of the PCB -- Mathematical Solution

This method uses a totally different idea which was already discussed in chapter 6.2. (fundamentals of patch antennas). So we must have a look at the nice figure concerning „current / voltage / impedance values on a patch“:

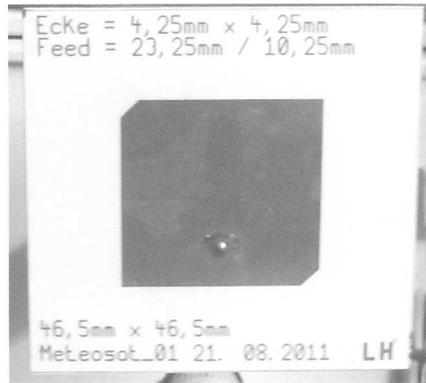
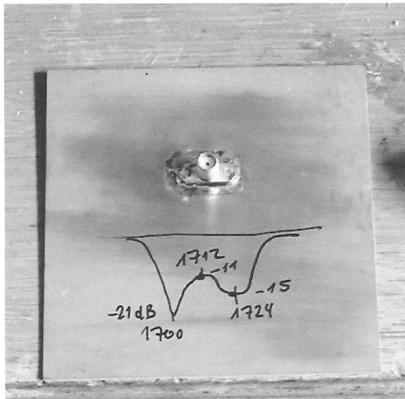


a) At the patch centre the voltage crosses Zero but the current shows a maximum value. This gives an impedance of 0Ω .

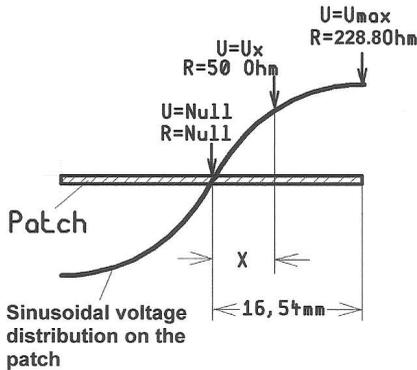
b) At a radiating edge of the patch the current amplitude is very small, but the voltage is at its maximum value. This gives a high input impedance value of more than 100Ω (in our previous example: 209Ω , but the exact value is 228.8Ω).

So somewhere between these two points must exist the “perfect matching to 50Ω ”.

We only have to drill a small hole at this point and then connect this point to the centre conductor of an SMA socket (which is soldered to the lower side at the “infinite ground plane”). The following two pictures show the realization of this idea for a 1700MHz Meteosat antenna.



But how can this mystic point be found?



It is important to know that such a transmission line also acts as an impedance transformer.

But remember:
The **voltage ratio** of a transformer is equal to the winding ratio.

The **impedance ratio** varies with the square of the winding ratio!

So we can write:

$$\left(\frac{U_x}{U_{MAX}}\right)^2 = \frac{50\Omega}{228.8\Omega}$$

or:

$$\frac{U_x}{U_{MAX}} = \sqrt{\frac{50\Omega}{228.8\Omega}} = 0.4675$$

Using the sinusoidal voltage distribution on the patch gives

$$\frac{U_{MAX} \cdot \sin\left[\left(\frac{X}{16.54mm}\right) \cdot 90^\circ\right]}{U_{MAX}} = 0.4675 = \sin\left[\left(\frac{X}{16.54mm}\right) \cdot 90^\circ\right]$$

Mathematics knows the „inverse sine function“ and this function calculates an angle of **27.87 degrees** for a sine value of 0.4675 using a good pocket calculator.

With

$$\frac{X}{16.54mm} = \frac{27.87^\circ}{90^\circ}$$

we get

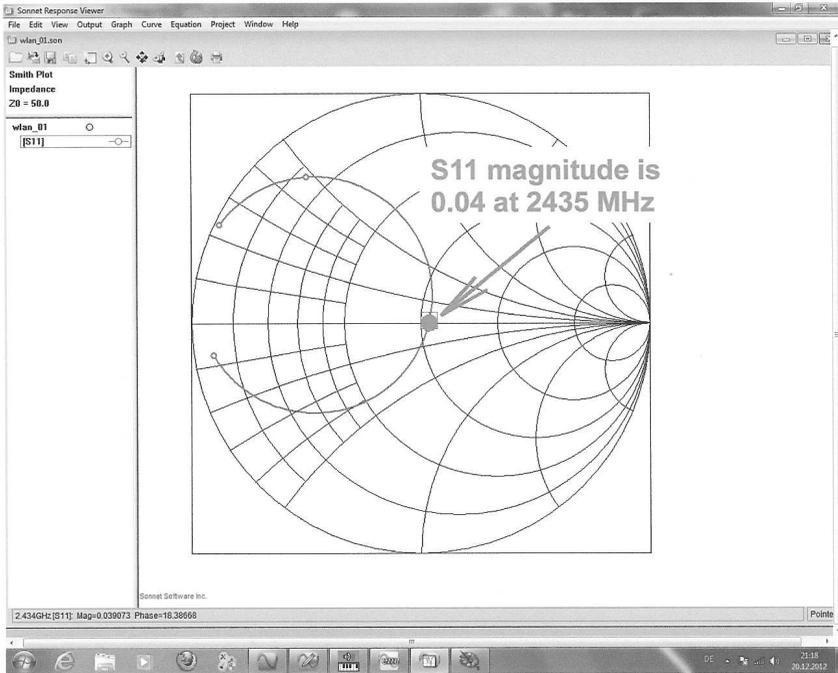
$$X = 16.54mm \cdot \frac{27.87^\circ}{90^\circ} = 5.12mm$$

for the distance between the patch centre and the feed point.

To check that this result is correct a SONNET LITE simulation was started for a complete simulation of the patch antenna including all losses.

The connection between the matching point and the central conductor of the SMA socket (soldered to the lower PCB side) was included as a **“via with a diameter of 0.8mm”**.

The simulation result is really interesting:



The via can be regarded as an inductor and decreases the resonant frequency from 2450MHz to 2435MHz but the magnitude of S11 is only 0.04. That confirms our PUFF work and it is time to produce a prototype.

But: would you now like to install SONNET LITE? Download the software from the Web and my free Sonnet Lite tutorial from my homepage (www.gunthard-kraus.de)



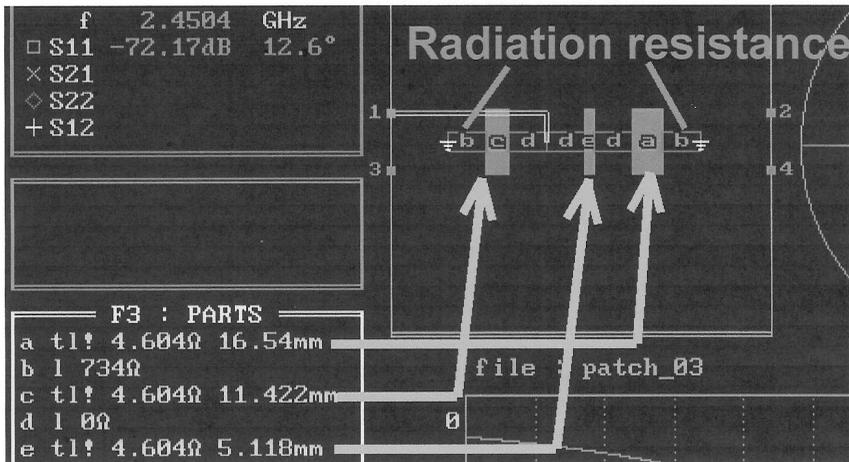
6.4.2.3. Coaxial Feed at the Lower Side of the PCB -- PUFF Solution

Now let us test the mathematical simulation result using PUFF:

Imagine cutting the patch antenna with a length of 33.08mm into two equal pieces then cut the left hand piece into two more pieces. The three line pieces have the same width and the same characteristic impedance of 4.604Ω (as designed in chapter 6.4.1)

The electrical length of the right piece (from the centre to the right patch end) is 90 degrees at 2450MHz (= 33.08mm / 2 = 16.54mm).
 The remaining left line piece is also cut in two pieces. The piece in the middle must have a length of 5.12mm (= Sonnet Lite simulation) and the most left a length of 16.54mm – 5.12mm = 11.42mm. These two pieces are now connected together by a short circuit bridge and port 1 is connected to this point.

Now press F1 and draw this layout. Do not forget the radiation resistance (each 734Ω) at the left and at the right patch edge. Use this figure as specimen:



Now a simulation can be started using the following settings:

Design frequency (In Window F4) = 2.45GHz

1000 points

Smith radius = 0.01

Frequency range = 2.44 to 2.46 GHz

Minimum value at the vertical diagram axis = -100 dB

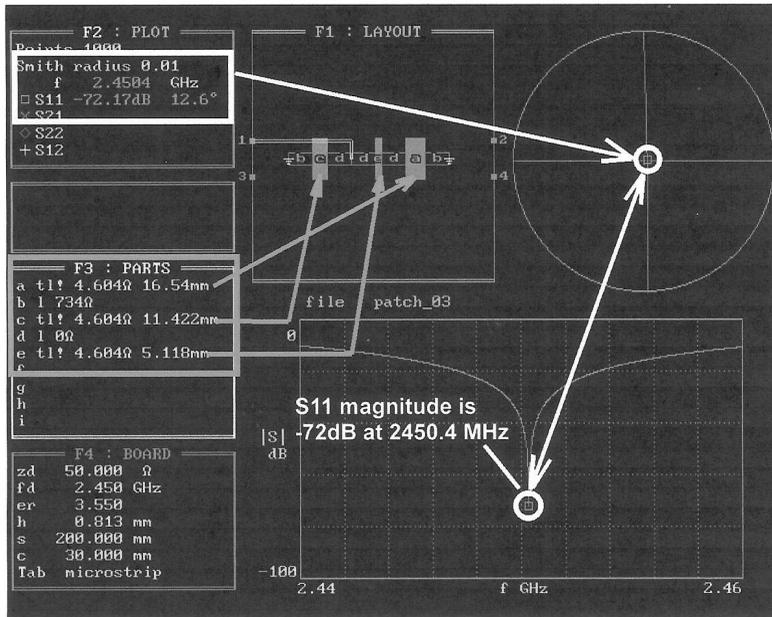


Handle the optimization in the following manner:

If the S11 curve is crossing the horizontal axis of the Smith chart at the left hand side of the centre then the input resistance is too low. Please increase the length of the line piece in the middle and decrease the left line piece length by the same amount. The sum of these two lengths must always be 16.54mm.

If the S11 curve is crossing the horizontal axis of the Smith chart at the right hand side of the centre then the input resistance is too high. Please decrease the length of the line piece in the middle and increase the left line piece length by the same amount. The sum of these two lengths must always be 16.54mm.

This is the end:



You should now stop with simulating and manufacture a prototype PCB. But compare this result to the quarter line matching procedure in the preceding chapter. The difference between the frequency for best matching and 2450 MHz is only 400 kHz. And an S11 magnitude value of -72dB will do the job.

I think: well done.



6.4.3. How to determine the Power Bandwidth?

This is a little difficult because instead of voltages you must think in „powers“. Incident and reflected waves are expressed in voltages but in reality powers are travelling on the line from the generator to the termination and vice versa. This „thinking in voltages“ is only necessary for easier calculation and measurement and better understanding.

If you now ask for the frequencies at which the radiated power has decreased by 30% you must at first start with 100% of radiated power at the resonant frequency and no reflection. This is due to a very small S11 value at this frequency.

A reduction of 30% in radiated power means that the missing 30% are reflected towards the generator and so the S11 value at this frequency must be

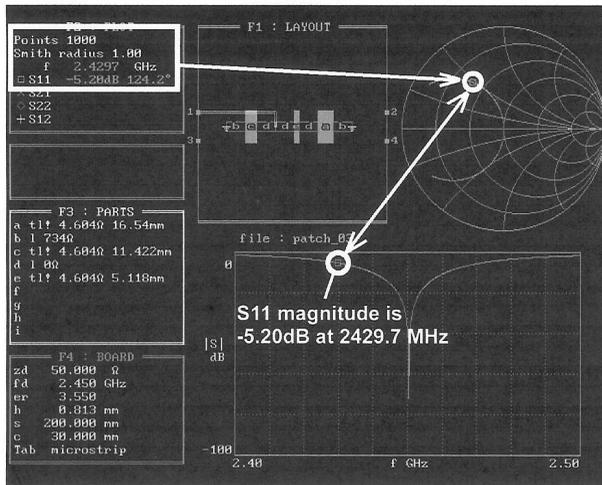
$$\frac{P_{\text{reflected}}}{P_{\text{incident}}} = \left(\frac{U_{\text{reflected}}}{U_{\text{incident}}} \right)^2 = 0,3 = r^2$$

This gives (when using a square root) a magnitude value for S11 = r = 0.55

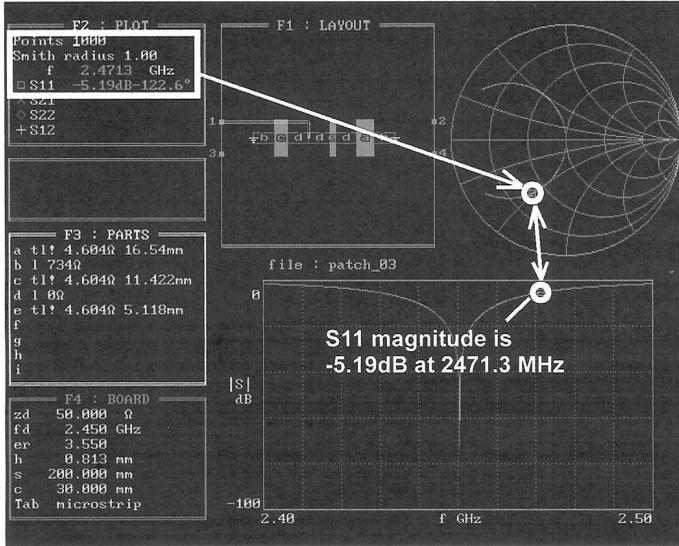
So move the cursor on the S11 curve (using the page up and the page down key) to the two points with a S11 magnitude = 0.55 = -19dB.

Note these two frequencies and calculate the difference.

This frequency difference is the “bandwidth for a -30% power reduction”.



Decreasing the frequency gives an S11 value of -5.2dB at 2429.7 MHz.....



...and increasing a frequency of 2471.3 MHz.

This gives a bandwidth of 41.6 MHz for a transmitted power reduction of 30%.



4.

Summary

It is a pleasure to be able to work properly with this RF calculator again. Modern microwave CAD programs with their huge model libraries are vastly superior in principle of course.

When it comes to testing out spontaneous ideas or the fast calculation of a single S Parameter, a calculator with simplicity, speed of input and operation has to be the first choice.

Therefore you should have PUFF as well as powerful simulation programs (the author has: ANSOFT Designer SV and Sonnet Lite) installed at the same time on your PC or your notebook. Have fun with PUFF and RF technology to produce countless new circuit ideas.

5.

References

[1] The author's web site: www.gunthard-kraus.de

[2] The new CD can be purchased from VHF Communications Magazine for £26. Web site: www.vhfcomm.co.uk

Amateur Television Quarterly

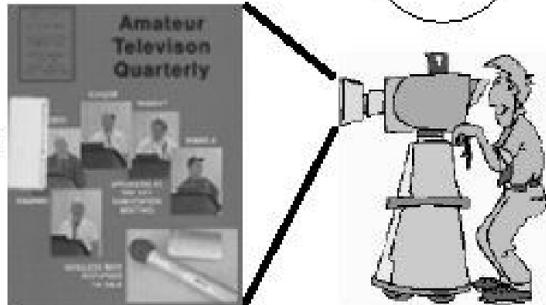
Great articles on :

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

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

*Don't miss another issue.
Subscribe Today!*

VISA
M/C
AMEX



Write or check webpage TODAY for more information!

Mike Collis, P.O. Box 1594, Crestline, CA 92325

<http://atvquarterly.com>



Operational Transconductance Amplifiers (OTA)

The question is: what is an Operational Transconductance Amplifier. That is answered in this article and with the help of LTspice the basic circuit design, as well as some applications are described. The LT 1228 by Linear Technologies that is already in the LTspice component library is used for the examples.

1.

Introduction

The comparison between the Operational Transconductance Amplifier (OTA) and conventional operational amplifier (OP) can be made by looking at the main features of the OTA as well as similarities and differences. The OTA has a high impedance differential input with no input current just like a conventional operational amplifier. Both types of amplifier have a differential input. In contrast to the OP that has a low impedance output, the OTA has a high impedance output. The OTA is a voltage source (Fig 1).

For the ideal OP ($R_i = 0$) so that the terminal voltage U at the output is the same as the open circuit voltage $U_o = A(U_p - U_n)$. The current output from the OP depends on the load $I = U_o/R_L$. If the

internal resistance of the voltage source does not equal zero the current flow causes a voltage drop across R_i . The Terminal voltage U is then less than the open circuit voltage and load dependent. An ideal voltage source must not be short circuited!

The short circuit current (I_o) of the OTA is the product of the transconductance (g_m) of the amplifier and the differential voltage at the inputs: $I_o = g_m (U_p - U_n)$. Ideally, if the internal resistance R_i of the current source is infinitely large, the output current is equal to the short circuit current. The voltage at the output is a voltage drop across the load impedance $U = I_o \cdot R_L = g_m \cdot R_L \cdot (U_p - U_n)$. The greater the load, the greater the output voltage. The output should not be left open circuit but can be short circuited.

The product $g_m \cdot R_L$ corresponds to the voltage gain of the OP.

The transconductance (g_m) can be controlled by an external current I_{ABC} (Amplifier Bias Current). This can be used to:

- control the gain of a circuit
- match a filter circuit cut off frequency

The ability to control the transconductance and the current output make the OTA an attractive building block in the development of active filters. However, the

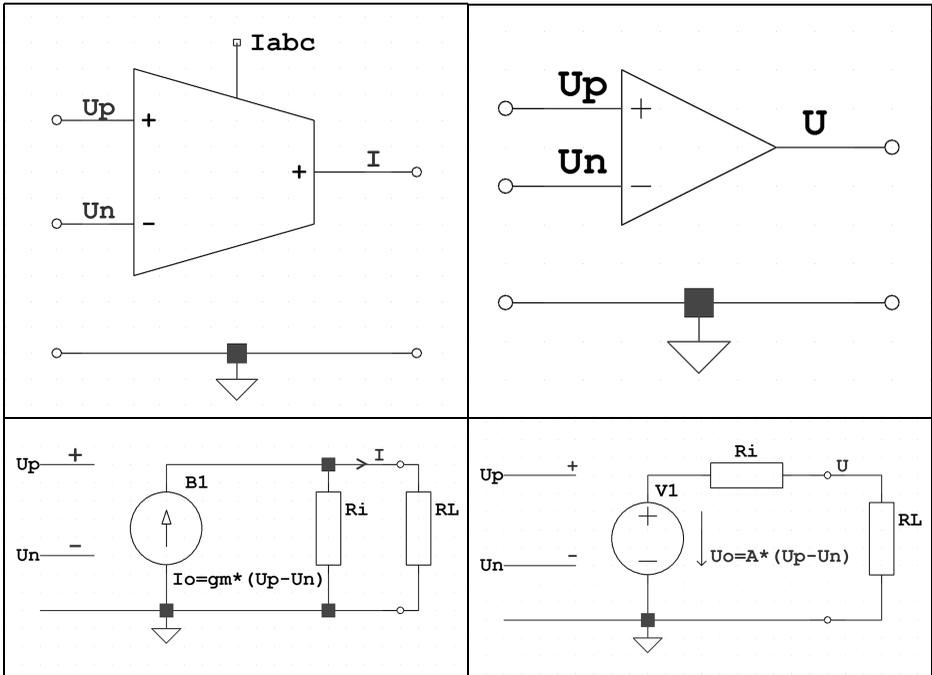


Fig 1: Comparison between the OTA and conventional Operational amplifier (OP).

OTA is particularly important for integrated analog filters, the so-called OTA-C or gm-C-Filter [7] that are composed of Transconductance Amplifiers and capacitors.

2.

Circuit of a Transconductance Amplifier

The differential amplifier and the current mirror are the basic building blocks for analog integrated circuits. The functioning of the current mirror should be understood in order to understand the function of the Transconductance Amplifier.

2.1. The current mirror

The current mirror is a specialised form

of a current source [1, 3]. Its output current is equal to an adjustable reference current. Fig 2 shows two simple circuits that use NPN or PNP transistors. Both transistors must work in their active area, i.e. in the horizontal part of the transistor characteristic $I_C = f(U_{CE}, I_B)$. The transistor Q1 is connected as a diode. The base collector voltage U_{CB} is zero, i.e. the base-collector-diode is short circuited, but it is in the active area. The base-emitter diode is conducting.

The base-emitter voltage U_{BE} is the same for both transistors and therefore the same collector currents flow. In the simple circuits shown in Fig 2 the collector currents are slightly different because the current I_{REF} must provide the base currents of both transistors. The ratio of the two currents can be shown to be [1]:

$$S = \frac{I}{I_{ref}} = \frac{B}{B+2}$$

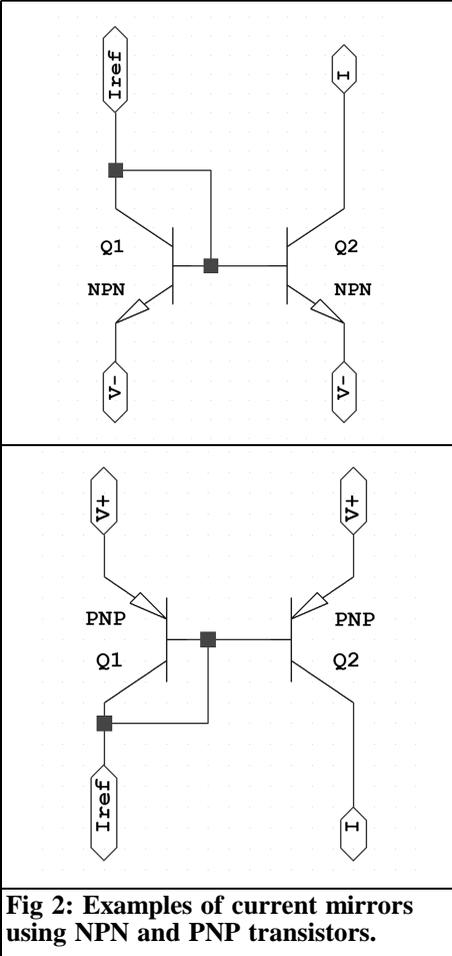


Fig 2: Examples of current mirrors using NPN and PNP transistors.

For a current gain $\beta = 100$ the difference is less than 2%.

The literature [1, 3] contains improved current mirror circuits such as the Wilson circuit used in the circuit of an OTA.

2.2. Hierarchical blocks

To simplify using the current mirror circuits in later applications they are “packaged” in hierarchical blocks. A hierarchical block is a part of a circuit that is assigned a symbol. The symbol is used as a building block in a parent circuit. The building block has a net list and a circuit.

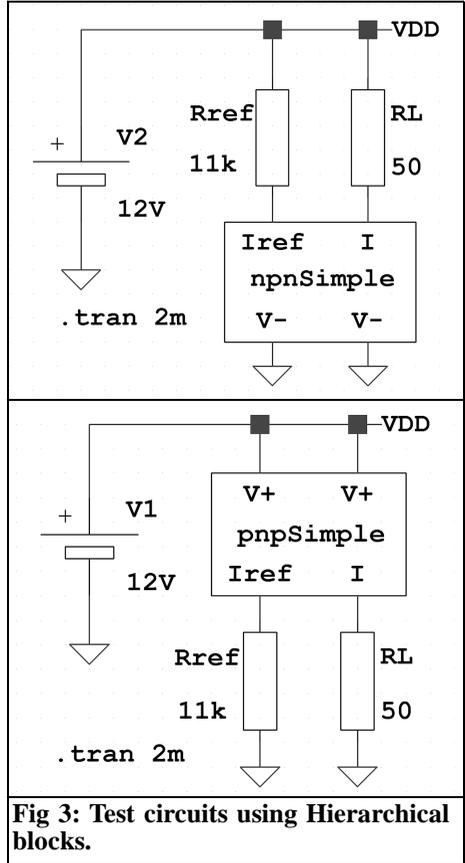


Fig 3: Test circuits using Hierarchical blocks.

2.2.1. Structure

Hierarchical blocks can be easily created with LTspice (Online help, Schematic capture, Hierarchy). The method will be briefly outlined: first, create the desired part as an LTspice circuit and name the connections. The "F4" key opens the "net name" dialog; the name and type (port type) are entered here. Store the circuit with a meaningful name (*.asc) in the current working directory. The first part is done. Now from the "Hierarchy" menu, open the dialog for creating a new symbol (create a new icon). Create the glyph and store it in the same working directory with the same name as the drawing (*.asy). The symbol type is "Block"; other attributes are not needed. To use the hierarchical block open the

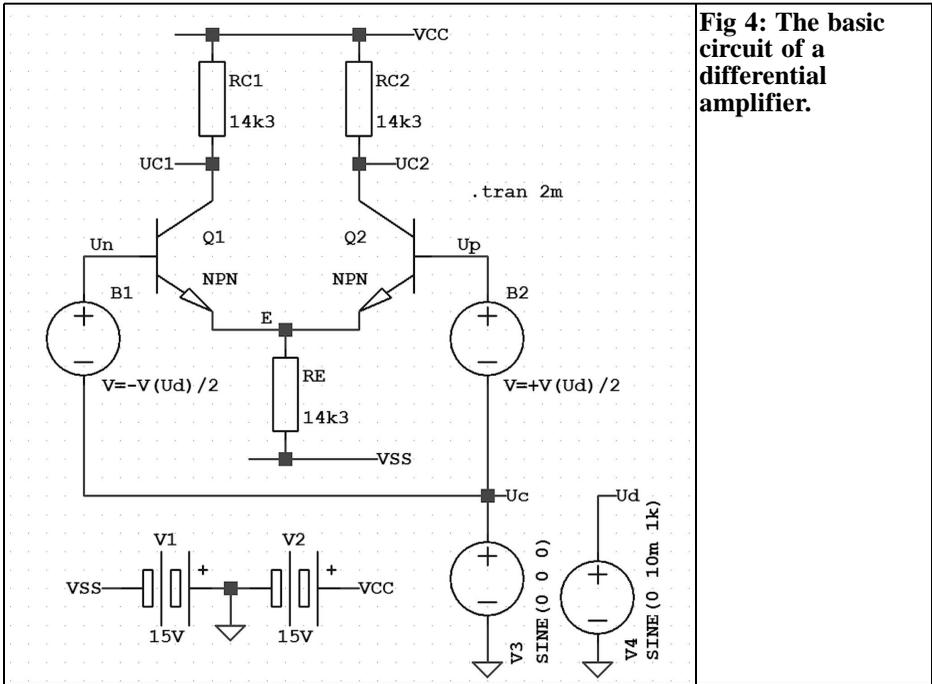


Fig 4: The basic circuit of a differential amplifier.

standard dialog as usual "select component symbol" (press F2) and select the desired symbol from the current working directory (top directory: c:\...), but not from the LTspice library.

2.2.2. Test Circuits

The test circuits that are shown in Fig 3 demonstrate the application of hierarchical blocks and check the function of the corresponding current mirror circuits. The reference current is set by a resistor. The current through the load resistor R_L is equal to the reference current. The user can vary the resistance R_L in the LTspice simulation to check that the load current I is largely independent of R_L as long as the transistors are in their active region. Clicking with the right mouse button on the hierarchical block opens the dialog "navigate/edit schematic block". From there the icon or the drawing and editing can be opened. Voltages and currents within the hierarchical block can be displayed if the corresponding LTspice options (Tools, Control Panel, save defa-

ults) are selected.

2.3. The differential amplifier

The basic circuit of a differential amplifier is shown in Fig 4. This circuit, with the resistors replaced by a current mirror, is used for the input circuit of Operational Amplifiers, Transconductance Amplifier and other analog integrated circuits.

2.3.1. Common mode and differential behaviour

The operation of the differential amplifier to differential as well as common mode inputs can be tested. For differential input U_n and U_p have equal large amplitudes but a phase shift of 180° . For common mod U_n and U_p are the same size and in phase. An independent source V4 controls the voltage U_d of the dependent voltage sources B1 and B2 with voltages of $(-U_d/2)$ and $(+U_d/2)$. The dependent voltage source is available in the LTspice library under "bv" (arbitrary behavioural voltage source). The independent source



V3 provides the common mode voltage U_c . The sum of differential voltage ($\pm U_d/2$) and the common mode voltage U_c acts on the base of the transistors. The common mode behaviour is not discussed here. The V3 source is set to 0V.

2.3.2. Operating point

First of all, the working point of the amplifier must be set. The operating point is set at $U_n = U_p = 0V$ ($U_c = 0, U_d = 0$). The current through the common emitter resistance R_E is arbitrarily selected as $I_{RE} = 1mA$. This current is the I_{ABC} current used to control the transconductance. The base currents are ignored. The emitter of Q1 and Q2 will be at a voltage of about 0.7V below their bases with the base-to-emitter diodes conducting. With a supply voltage of $\pm 15V$ the voltage across R_E is 14.3V. Thus $R_E = (15 - 0.7)V/1mA = 14.3k\Omega$. The current is shared equal between both transistors so $I_{C1} = I_{C2} = I_C = 0.5mA$. The collector resistors $R_{C1} = R_{C2} = R_C$ is selected so that the collector voltage at the operating point of approximately half the positive supply voltage $U_{C1} = U_{C2} = 15V - 14.3k\Omega \cdot 0.5mA = 7.85V$. Thus the dynamic range is at its maximum. When $U_{C1} = U_{C2}$ the differential output voltage is 0V ($U_{C1} - U_{C2}$).

With the LTspice currents and voltages at the operating point determined, the analysis type ".OP" returns a table with all voltages and currents. You can see individual voltages and currents at the operating point in the status bar if you hold the cursor over the appropriate node or component in the circuit. The results of the simulation are: $U_{BE} = 0.756V$ instead of 0.7V, $U_{C1} = U_{C2} = 7.948V$ instead of 7.85V.

2.3.3. Differential operation

U_d is selected for the differential voltage of 10mV [sine(0 10m 1k)]. It is well within the linear range of the amplifier. The voltages U_n and U_p have half the amplitude and are equal. According to the law of superposition, the individual

currents and voltages can be determined for each source and the subtotals added. First, the base of Q2 is held at ground; however it cannot be connected directly to ground otherwise the simulator complains that the sources V3 and B2 would be connected in parallel. It can be achieved by setting the source B2 "V = 0": U_c is already zero. A 5mV sine wave signal is fed to the base of Q1. The transistor Q1 works in common emitter with current feedback while Q2 in common base with an upstream emitter follower Q1. The feedback resistor for Q1 consists of the parallel connection of R_E and the input resistance Q2 base circuit. This is r_{BE}/β and very much smaller than R_E therefore R_E in parallel can be ignored. The voltage gain of Q1 is:

$$v = \frac{\beta (R_C \parallel r_{CE})}{r_{BE} + (\beta + 1)R_E} \approx \frac{\beta}{2 r_{BE}} R_C$$

with:

$$r_{be} = \frac{U_T}{I_B}, \quad I_B = \frac{I_C}{\beta}$$

and $U_T = 26mV$, the temperature voltage at room temperature.

It follows that:

$$v = \frac{I_C}{2 U_T} R_C$$

If the collector current in equations above is replaced by $I_{RE} = I_{ABC}$ the transconductance can be expressed in relation to the control current. With $I_c = I_{ABC}/2$ the voltage gain of the transistor Q1 is :

$$v = \frac{I_{ABC}}{4 U_T} R_C$$

Using the values for I_{ABC} and R_C from Fig 4 gives $v = 137.5$.

Transistor Q2 is in common base. The gain of a common base circuit is specified in [3] for the case when the signal source has an internal resistance of 0Ω. This is:



$$v' = \frac{\beta}{r_{BE}} R_C$$

The upstream emitter follower is the signal source for Q2 with the internal resistance r_{BE}/β . The input resistance of the base circuit is also r_{BE}/β . The voltage gain v' is smaller by a factor of 2, the voltage division of two equally value resistors. Finally:

$$v = \frac{\beta}{2 r_{BE}} R_C$$

exactly the same as that of the transistor Q1.

By analogy the same is true for the second branch with $U_n = 0$, and $U_p = U_d/2$. No matter which transistor is considered the gain in both branches are controlled. The phase conditions are as follows. Because Q1 is an emitter follower the signal on its collector is the same as the input signal. The signal on the collector of Q2 is in phase with the input because neither the emitter follower nor the common base circuit change the phase. For both transistors the signal on its own collector is the opposite phase to its base signal (negative reinforcement) and to the collector of the other transistor (positive reinforcement). The signals U_p and U_n to both transistors can be added. The voltage gain of the differential amplifier is therefore:

$$v_D = 2 v = \frac{I_{ABC}}{2 U_T} R_C$$

The symmetrical output voltage is:

$$U = U_{C1} - U_{C2} = \frac{19.23}{V} I_{ABC} R_C (U_p - U_n)$$

Comparing the equation with the equation in the introduction $U = g_m R_L (U_p - U_n)$, means that you can specify the transconductance of the differential amplifier:

$$g_m = \frac{I_{ABC}}{2 U_T} = \frac{19.23}{V} I_{ABC}$$

The transconductance of the differential amplifier is proportional to the current I_{ABC} . It is independent of the collector resistance. The voltage gain is $v_D = g_m \cdot R_C$. With the values from Fig 4:

$$v_D = \frac{I_{ABC}}{2 U_T} R_C = 275$$

Example:

An example of the AC voltage generated at the collectors by using the symmetrical voltage U_d : LTspice can generate the voltages using time analysis: ".tran 2m" Using this for Fig 4:

Case a)

$$U_d = 10mV, U_n = -5mV, U_p = 0mV$$

$$Q1: U_{C1a} = (-137.5) (-5mV) = +687.5mV$$

$$Q2: U_{C2a} = (+137.5) (-5 mV) = -687.5mV$$

Case b)

$$U_d = 10mV, U_n = 0, U_p = +5mV$$

$$Q1: U_{C1b} = (+137.5) (+5 mV) = +687.5mV$$

$$Q2: U_{C2b} = (-137.5) (+5mV) = -687.5mV$$

Total voltage on the collector of Q1:

$$U_{C1} = U_{C1a} + U_{C1b} = +1.375V$$

Total voltage on the collector of Q2:

$$U_{C2} = U_{C2a} + U_{C2b} = -1.375V$$

Differential voltage:

$$U_o = U_{C1} - U_{C2} = 2.750V$$

The differential voltage can also be calculated from:

$$g_m = \frac{I_{ABC}}{2 U_T} = 19.23mS$$

$$v_d = g_m R_C = 19.23mS \cdot 14.3k\Omega = 275$$

$$U_o = v_d \cdot U_d = 275 \cdot 10mV = 2.75V$$

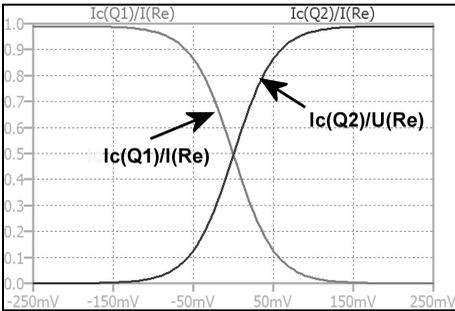


Fig 5: The transfer characteristic of the differential amplifier. It is only linear over an input range of $\pm 30\text{mV}$.

2.3.4. Transfer characteristic

The transfer characteristic for the differential amplifier is shown in Fig 5. To create this image, change the analysis type to "DC Sweep" in Fig 4:

```
.dc V4 - 250 m + 250 m 10 m.
```

This shows that the two currents $I_c(Q1)$ and $I_c(Q2)$ are related to $I(R_E)$ via the input voltage. The transfer characteristic clearly shows the non-linear behaviour of the differential amplifier. Linearity is only achieved over a narrow range of input voltage of approximately $\pm 30\text{mV}$. Furthermore it shows that the collector currents always add to total the current $I(R_E)$. When more current flows in one transistor, the current in the other reduces. The input voltage range can be extended by linearisation of the transfer characteristic. In Fig 4 this can be achieved by inserting a small resistor (e.g. 100Ω) in each emitter connection so that the transfer curve becomes flatter. The input voltage range is now approximately $\pm 100\text{mV}$. The price of this is a smaller transconductance (g_m).

2.4. The Transconductance Amplifier

A Transconductance Amplifier as shown in Fig 1 with differential (balanced) inputs and an unbalanced output can be realised if the emitter and the collector resistors are replaced by current mirror as shown in Fig 6. The transconductance

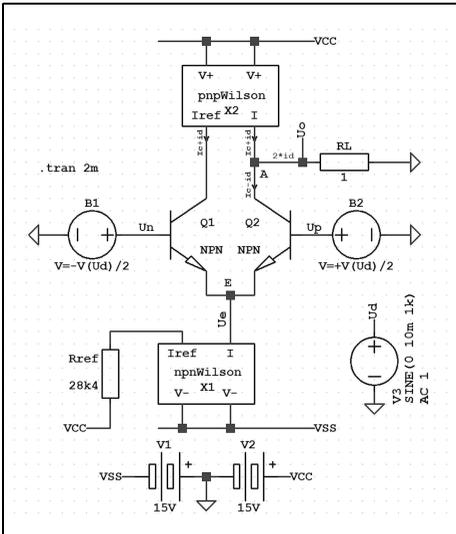


Fig 6: The Transconductance Amplifier. Only AC current flows through the load. This is magic. The transconductance is control by the current generated by R_{ref} .

(g_m) is only dependent on the DC current (I_{ABC}) at the operating point. This current is generated by the current mirror (X1) and feeds the emitters. The current mirror (constant current source) has a high resistance to current changes. The control current is set with the resistor R_{ref} .

The current mirror (X2) is connected with the reference side in the collector of transistor Q1 with the current mirror forming an active load for Q2. The load resistance R_L can be connected to the mirror side of the active load from the collector to ground. Both currents from Q1 and Q2 flow through the load.

A brief description of the active load is as follows:

First of all, the currents at the operating point will be considered. Both inputs are connected to ground ($U_c = 0, U_d = 0$). The currents through Q1 and Q2 are the same (e.g. $I_c = 0.5\text{mA}$) and in phase. The collector current of Q1 is the reference current flowing through the active load.

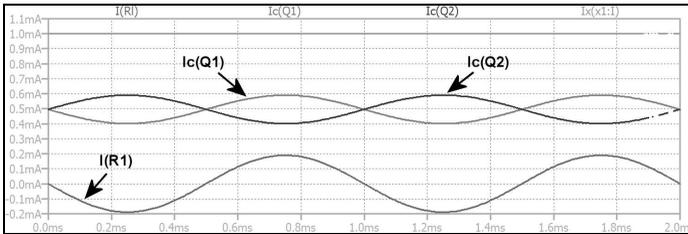


Fig 7: The currents flowing in the Transconductance Amplifier.

On the mirror side of the current mirror the output current flows out of node A. The collector current of Q2 flows away from node A so the current total is equal to zero. The currents cancel out so the load current is zero.

When a signal is applied the amplitude of the alternating current through Q2 is equal to twice the alternating current through Q1. In the two branches the current consists of an in phase DC part I_c and an anti-phase AC part I_d . At node A the sum of the currents is $(I_c + i_d) - (I_c - I_d) = 2 \cdot I_d$. The DC currents cancel out again and the alternating currents add up to the

total AC current flowing through the load. Only AC current flows through the load without using a decoupling capacitors!

Fig 7 shows the current flows in the Transconductance Amplifier. The current $I_x(x1:I) = I_{ABC}$ is 1.0mA. The DC component is $495\mu A$ for both $I_c(Q1)$ and $I_c(Q2)$. The base current is $5\mu A$. The amplitudes of the currents are $i_{RC1} = i_{RC2} = 94.5\mu A$. The alternating current through the load has an amplitude of $i_{RL} = 2 \cdot 94.5\mu A = 189\mu A$. The formula does not take into account the base currents therefore gives a slightly higher value:

$$\hat{i}_o = g_m I_{ABC} (U_p - U_n) = \frac{19.23}{V} \cdot 1mA \cdot 10mV = 192.3\mu A$$

```
* OTA1
* SUBCIRCUIT CONNECTIONS IN+ IN- I+
.SUBCKT OTA1 1 2 3
* Bnnn n001 n002 I=<expression>
B1 0 3 I={gm}*V(1,2)
.ENDS OTA1
```

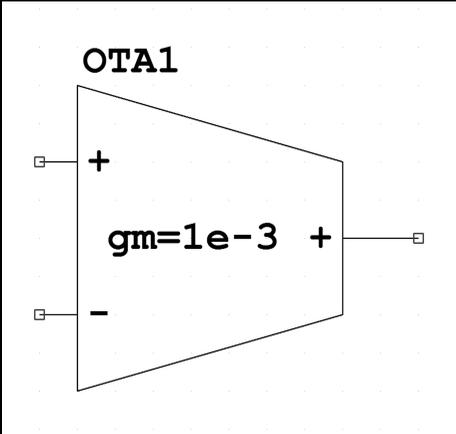


Fig 8: The net list and symbol for a Transconductance Amplifier. The transconductance must be manually set using the formula $I = g_m(U_+ - U_-)$.

3. OTA models

In the literature you will find various types of Transconductance Amplifier, their output can be balanced or unbalanced. The OTAs with unbalanced output have the output current direction defined (current source or current sink). The circuits in this article use OTAs with unbalanced current source outputs. The symbol for this OTA has a + sign on the output (a current sink has a - sign). The net list and symbol for an ideal OTA is shown in Figs 8 and 9. The equivalent circuit is shown in Fig 1. The internal resistance R_i is infinite. The ideal voltage controlled current source is defined in the



```
* OTA1c
* SUBCIRCUIT CONNECTIONS IN+ IN- I+ Iabc
.SUBCKT OTA1c 1 2 3 4
* Bnmn n001 n002 I=<expression>
R1 4 0 1
B1 0 3 I=19.23*I(R1)*V(1,2)
.ENDS OTA1c
```

Fig 9: The net list and symbol for a Transconductance Amplifier with a control input. The calculations are: $g_m = 19.23 \cdot I_{ABC}$ and $I = g_m(U_+ - U_-)$.

network list line

$$"B1 0 3 I = \{g_m\} * V(1,2)"$$

(the online help shows the circuit element B is an arbitrary behavioural voltage or current source). The input voltages are connected to the positive and negative input of the OTA (pins 1 and 2). The transconductance gm is used in the model (in curly braces) as a parameter. The default of value for the symbol is defined using the symbol editor by selecting the "edit attributes" menu (or "Ctrl + A").

As an example enter " $g_m = 1e - 3$ " in the "SpiceLine". This will make it possible to change the value for each OTA later in the circuit. The second model shown in Fig 9 is the same type of OTA but an I_{ABC} input is provided. This model can be used as a signal multiplier (mixer, modulator, etc.). It is also used in the circuit of an AM modulator shown in the datasheet for the LT1228 on p. 22 [8].

```
.tran 2m
R=1/gm=1k
I=10mV/1k=10uA
```

Fig 10: An OTA used as an active resistor.

4.

Basic circuits using OTAs [4]

When you start to use conventional operational amplifiers you start with simple circuits e.g. summing amplifiers, inverting and non-inverting amplifiers and integrators. The same circuits can be tested with the OTA. Because they do not require feedback they are more stable and some circuits will be easier. Another major difference is the greater bandwidth of the OTA compared to the OP.

4.1. Active resistance

The OTA circuit shown in Fig 10 operates as a resistor to ground. This can be shown very easily. At the current node the sum of all currents must be zero:

$$I + I_O = 0$$

$$I_O = -g_m U$$

$$I = g_m U.$$

The source "sees" the resistance

$$R = U/I = 1/g_m$$

Positive and negative resistors with respect to ground and "floating" resistors can be simulated with an OTA.

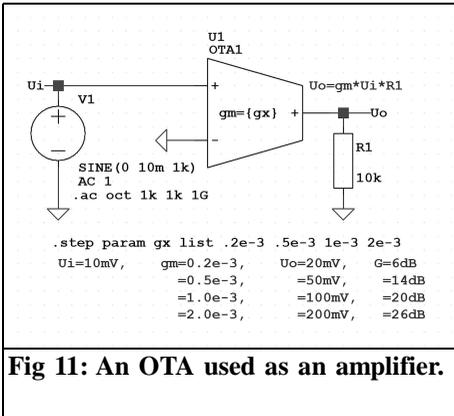


Fig 11: An OTA used as an amplifier.

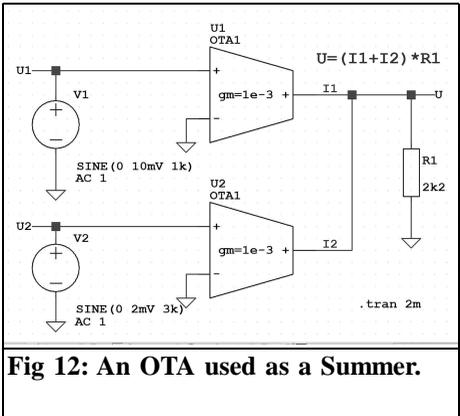


Fig 12: An OTA used as a Summer.

4.2. Amplifier

An output voltage can be generated from an OTA as a voltage drop from the output current through a resistor as shown in Fig 11. This can be the active resistance as above. With the input to the + input gives a non-inverting amplifier. If the inputs are swapped over it gives an inverting amplifier. The current I_{ABC} must be set for a real circuit. In a simulation with the model OTA1 the desired g_m value is entered manually. To do this use a right mouse click on the OTA symbol above the current value e.g. "gm = 1e - 3". The new value can be entered in the dialog box that opens. The influence of the transconductance is shown in Fig 11 by simulating the gain for several g_m values. The values used {gx} are contain

ned in the variables list (.step statement).

4.3. Summers

The summation of two voltages is controlled by the sum of the currents of the load resistor. The circuit is shown in Fig 12 with the output shown in Fig 13. By setting the transconductance the gain for each valued to be summed can be adjusted individually:

$$U = (I_1 + I_2)R_L = (g_{m1} U_1 + g_{m2} U_2) R_L$$

4.4. Integrator

Current sources are ideally suited for integration. Fig 14 shows the circuit with a capacitor as a "load accumulator" The frequency response of the integrator is shown in Fig 15. The transfer function is very easy to determine:

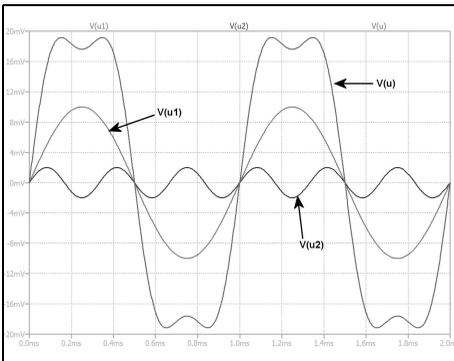


Fig 13: The inputs and outputs of the Summer shown in Fig 12.

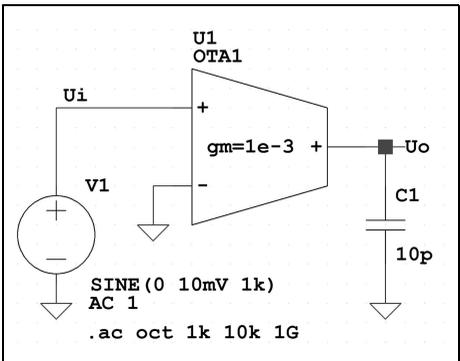


Fig 14: An OTA used as an Integrator.

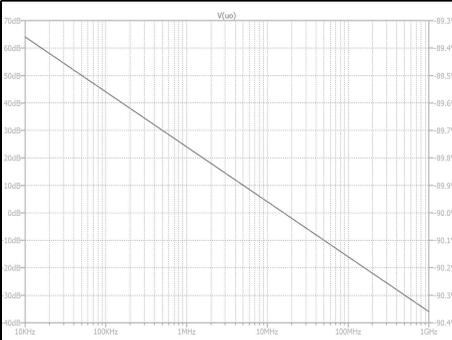


Fig 15: The frequency response of the Integrator shown in Fig 14.

$$U_o = I_o Z_L = g_m U_i \frac{1}{sC}$$

$$\frac{U_o}{U_i} = \frac{1}{s} \frac{g_m}{C}$$

The cut-off frequency is:

$$\omega_o = g_m / C$$

and

$$f_o = g_m / 2\pi C$$

At this frequency the input and output voltage are the same (gain 0dB). The cut-off frequency is proportional to the transconductance that can be adjusted with the control current I_{ABC} . With the values from Fig 14 $f_o = 15.9\text{MHz}$.

It is interesting that the capacitor retains its charge when the input signal is zero.

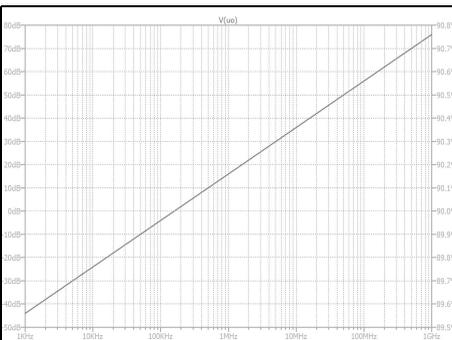


Fig 17: The frequency response of the Differentiator shown in Fig 16.

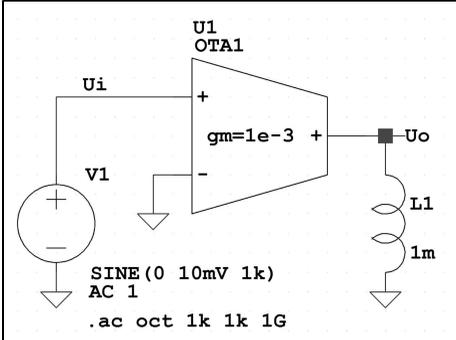


Fig 16: An OTA as a Differentiator.

The capacitor can discharge slowly into the high output impedance of the current source. Fig 15 shows the frequency response of the Integrator.

4.5. Differentiator

The differentiator is just as easy, a coil is used as the load. Fig 16 and 17 show the circuit and the frequency response of the differentiator. The transfer function is easy to define:

$$U_o = I_o Z_L = g_m U_i sL$$

$$\frac{U_o}{U_i} = s g_m L = \frac{s}{\omega_o}$$

The cut-off frequency is:

$$\omega_o = 1/g_m L$$

and

$$f_o = 1/2\pi g_m L$$

At this frequency the input and output voltage are the same (gain 0dB). With the values from Figure 16 the cutoff frequency is $f_o = 159.1\text{kHz}$.

The next section shows how an inductor can be simulated with an OTA so the differentiator can be re-designed with an active inductance.

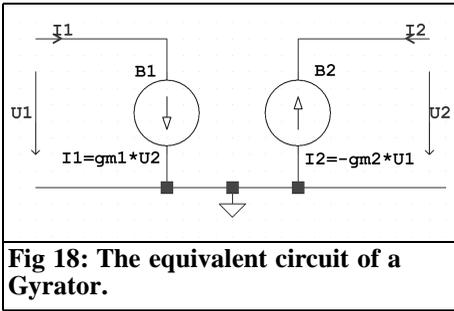


Fig 18: The equivalent circuit of a Gyrator.

4.6. Active inductance

Different circuits have been developed to realise an active inductance. At low frequencies the so-called Generalised Impedance Converter (GIC) is common. [6]. The GIC transforms a given impedance according to a selected function. It is built with operational amplifiers.

The Gyrator is a special impedance converter. It was proposed in 1948 as a hypothetical fifth theoretical element in addition to the four electrical passive components: resistor, coil, capacitor and transformer [2]. It can invert an impedance so that a capacitor can act like an inductor. The Gyrator is very simple to set up with OTAs.

The definition of a gyrator in admittance form is given by the four parameters:

$$\begin{pmatrix} I_1 \\ I_2 \end{pmatrix} = \begin{pmatrix} 0 & g_1 \\ -g_2 & 0 \end{pmatrix} \begin{pmatrix} U_1 \\ U_2 \end{pmatrix}$$

These equations describe two voltage controlled current sources (Fig 18). The

negative sign is taken into account by the reversal of a source.

This can be realised with two OTAs that are voltage controlled current sources with one having the positive input active and the other with the negative input active (Fig 19).

With:

$$dety = y_{11}y_{22} - y_{12}y_{21} = g_1g_2$$

and

- Y_S the source admittance
- Y_L the effective admittance
- Y_{in} the input impedance Z_{in}
- Y_{out} the output impedance Z_{out}

Then:

$$Y_{in} = \frac{1}{Z_{in}} = \frac{I_1}{U_1} = \frac{dety + y_{11}Y_L}{y_{22} + Y_L} = \frac{g_1g_2}{Y_L}$$

$$Y_{out} = \frac{1}{Z_{out}} = \frac{I_2}{U_2} = \frac{dety + y_{22}Y_S}{y_{11} + Y_S} = \frac{g_1g_2}{Y_S}$$

The input admittance is inversely proportional to the output admittance of the four pole network i.e. the Gyrator with a capacity on the input is "seen" as an inductance.

The example Fig 19 shows a circuit of a Gyrators to reproduced an inductance. The equivalent passive circuit is shown as an example. The frequency response is shown in Fig 20. The impedance at the

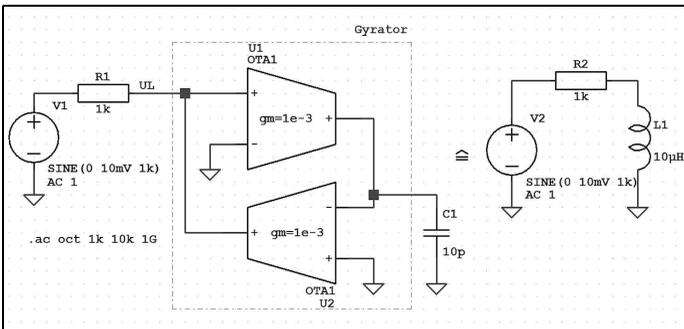


Fig 19: Two OTAs used as a Gyrator

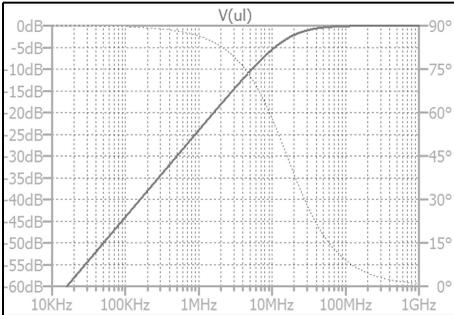


Fig 20: The frequency response of the Gyrator shown in Fig 19.

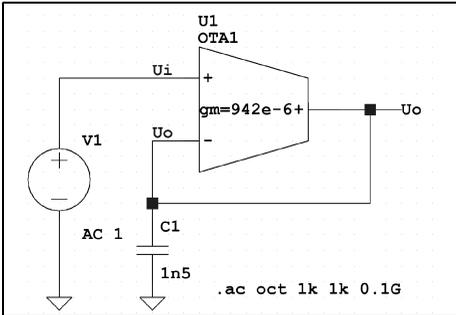


Fig 21: An OTA used as a first order low pass filter.

input of the OTA (node U_L) is:

$$Z_{in} = \frac{1}{g_{m1} g_{m2}} sC = s \frac{C}{g_{m1} g_{m2}} = sL$$

The inductance $L = C/g_{m1}g_{m2}$ can be adjusted using the transconductance g_{m1} or g_{m2} .

Example:

With the values from Figure 19, the inductance is:

$$L = \frac{10 \text{ pF}}{1 \text{ mS } 1 \text{ mS}} = 10 \mu\text{H}$$

For $R_1 = 1\text{k}\Omega$ the cut-off frequency is:

$$f = \frac{R}{2\pi L} = \frac{1\text{k}}{2\pi 10 \mu\text{H}} = 15.91 \text{ MHz}$$

For comparison the frequency response of a passive R_L circuit can be simulated and both frequency response curves are identical.

4.7. First order low pass filter

The methods for development of OTA filters are described in detail in [5] and [7].

A simple filter example is described below. A first order low pass filter with an external capacitor is shown in Fig 21.

The calculation of the transfer function is derived from the equation defining the OTA. From this:

$$I_o = g_m (U_i - U_o).$$

The voltage across C1 is:

$$U_o = I_o \frac{1}{sC_1}$$

It follows that the current flow through C1:

$$I_o = U_o sC_1 = g_m (U_i - U_o)$$

The transfer function is:

$$\frac{U_o}{U_i} = \frac{g_m/C}{s + g_m/C}$$

This result is multiplied by the transfer function of a first order low pass filter:

$$H = K \frac{\omega_0}{s + \omega_0}$$

for $K = 1$ and $\omega_0 = g_m/C$.

Example:

A low pass filter with a cut off frequency $f_o = 100\text{kHz}$ has been designed using a capacitor $C = 1.5\text{nF}$ and a transconductance $g_m = 942\mu\text{S}$. Fig 22 shows the frequency response using an ideal OTA. The low pass filter using an LT1228 [8] is shown in Fig 23 with the frequency response in Fig 24. From the data sheet, the relationship between the transconductance g_m and I_{ABC} is controlled by the

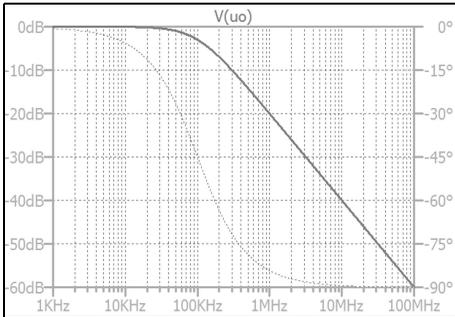


Fig 22: The frequency response of an ideal OTA used as a first order low pass filter.

current called I_{SET} :

$$g_m = \frac{10}{V} I_{SET}$$

The I_{SET} current can be adjusted by the voltage divider R1/R2 and the resistor R3. This and other ways to adjust the current I_{SET} can be found in the datasheet. The voltage divider R1/R2 is selected arbitrarily. With the resistors shown in Fig 23 it forms a voltage source with an open circuit voltage of 12.5V and an internal resistance 1kΩ. From the data sheet the I_{SET} input is defined by the forward voltage drop of two diodes more positive than the negative supply voltage i.e. a voltage of about -13.8V. The control current I_{SET} required is:

$$I_{SET} = \frac{g_m}{10/V} = 94.2\mu A$$

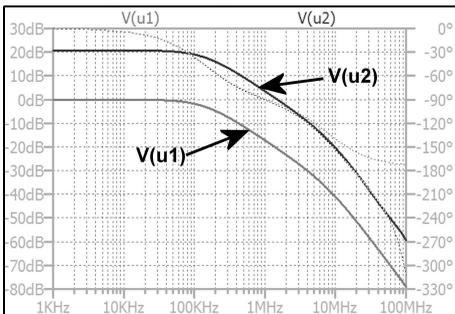


Fig 24: The frequency response of the low pass filter circuit shown in Fig 23.

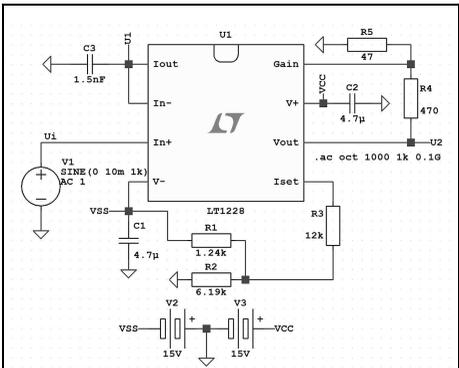


Fig 23: A first order low pass filter implemented with an LT1228.

Therefore the value of R3 is:

$$R_3 = \frac{(13.8-12.5)V}{94.2\mu A} - 1k\Omega = 12.8k\Omega$$

12kΩ was chosen

A more advanced circuit is shown in the datasheet (p. 17). The circuit has two inputs, a low pass filter input and a high pass filter input. To make a low pass filter, the high pass input is grounded and the low pass input driven. Conversely for a high pass filter the low pass input is grounded and the high pass input driven. If both inputs are driven, the result is an all-pass filter or phase shifter.

In addition to the OTA, the LT1228 also includes also a trans-impedance amplifier or Current Feedback Amplifier (CFA). In Fig 23 it is used to produce the output "V_{out}" that is a low impedance output. The signal On the "I_{out}" input to amplified by a factor of 1 + R4/R5.

5.

Summary

The circuit of a single-stage Transconductance Amplifier has been described above. Models of an ideal OTA were



created in a Simulator to study of circuits. Some typical circuits used with conventional operational amplifiers were tested with OTAs. Transconductance Amplifier are used mainly, but not exclusively, in the construction of the filters. You will find a wide range of applications in the data sheet for the LT1228 [8].

6.

Literature

[1] Integrierbare Grundsaltungen für analoge Signale, Th. J. van Kessel, R. J. van de Plassche, Philips Technische Rundschau, 32. Jahrg. 1971/1972, Nr. 1, S. 1-12

[2] The Gyrator, A New Electric Network Element, B. D. H. Tellegen, Philips Res. Rept., 3, 81-101 (Apr. 1948), <http://theestory.com/files/article-tellegen-gyrator.pdf>

[3] Halbleiter-Schaltungstechnik, Tietze/Schenk, Springer, 2002, Edition 12

[4] Active Filter Design Using Operational Transconductance Amplifiers: A Tutorial, R. L. Geiger, E. Sanchez-Sinencio, IEEE Circuits and Devices Magazine, Vol. 1, pp. 20-32, March 1985

[5] Active Filters, Theory and Design, S. A. Pactitis, CRC Press, 2007

[6] Aktive Filter und Oszillatoren, Lutz v. Wangenheim, Springer, 2008

[7] VLSI Analog Filters, Active RC, OTA-C, and SC, P. V. Ananda Mohan, Springer, 2013

[8] Linear Technology, LT1228, 100MHz Current Feedback Amplifier with DC Gain Control, <http://www.linear.com/>



Convention for Amateur Television 2013

- Technical talks program, with a special emphasis on "getting started in DATV"
- Demonstration area ranging from historic studio equipment to the latest DATV equipment
- Test and fix area
- Selected RF and video suppliers
- SMD soldering class
- Members bring and buy area

October 26th & 27th - Finningley Amateur Radio Club, Robin Hood International airport, Yorkshire.

See <http://www.batc.org.uk/cat13> for more details



Dirk Müller, DB6FM

The RTL-SDR Working with a USB stick

Part 2 continued from issue 2/2013

Good solutions for using SDRs across the spectrum between ultra long wave to SHF range have become simple and cost effective. This part of the article concentrates mainly on the different designs of the RTL SDRs and using the complete spectrum.

9.

New hardware features

In the short space of time since part 1 was published [4] a lot has happened.



Fig 1: The new shortwave converter by Frank Koeditz.

Firstly, various programs have been greatly improved, and secondly, a new short-wave converter for 0 to 55MHz (Fig 1) by Frank Koeditz [1] has become available. The various tuner models; the FC0013, E4000 and R829T have been measured and compared with each other.

The E4000 tuner is the first choice for me because it is the most compatible, it has a vast frequency range and the best integration with most programs especially HDSDR. The R820T tuner will hopefully soon be improved; there are still problems with the PLL lock. Furthermore the R820T was praised by some users as more sensitive, more on that later with the measurement results.

10.

SDR# new features

The new version of the SDR# now supports RDS that will surely inspire the FM DXer. Additionally the tuning has been simplified Fig 2. There are less numbers to deal with and the reception frequency can be "Configured" between the sidebands and the centre frequency. This eliminates the annoying LO spectral line in the earlier release when the centre and reception frequency were the same. The frequency can be set manually with-

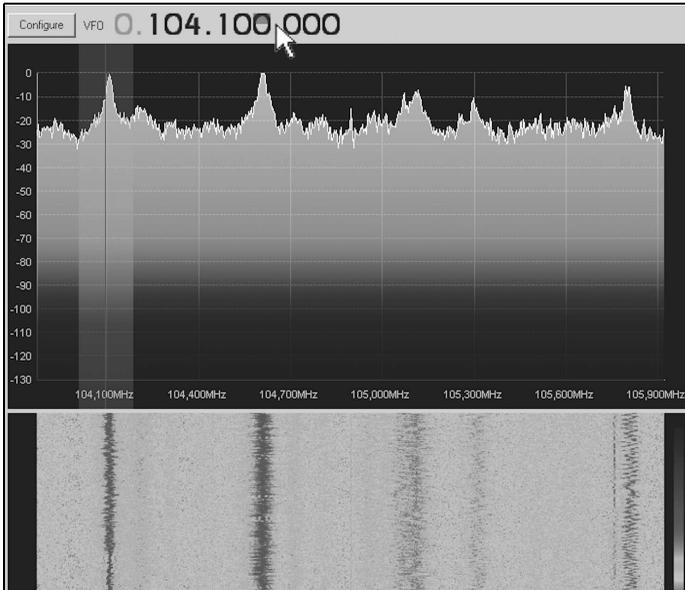


Fig 2: New version of SDR#: now with RDS and simpler tuning.

out the need to laboriously drag the mouse. The frequency can be entered or set with a click in the white frequency boxes, this is a beautiful innovation. There are more simplifications to the tuning. You can use the mouse pointer on the frequency, the number selected is coloured and the mouse wheel can be used to quickly scroll the frequency digit. A right mouse button click on a number quickly set it to "zero".

The default is now the frequency Manager. This allows fast jumping to an interesting frequency using internal frequency banks for a quick comparison of the conditions that can change quickly especially on shortwave. The enormous spectral range with a width of up to 2.4MHz over the selected shortwave range is very useful; other devices are not needed to locate interesting signals. You could not use the shortwave so comfortable and cheaply! If needed, you can always use an additional reference devices like an EKD300. The increased possibilities with new software and above all the shortwave converter are remarkable.

11.

The shortwave converter model 1205

A converter especially for the RTL-SDR is described in [1], the frequencies range of nearly 0MHz to 55MHz is covered with a 100MHz offset. Fig 3 shows an internal view of the converter. Thus the LW band is converted to use the optimum range of the RTL SDRs. The neat implementation of the frequency range means that the receive frequency can be calculated easily if the software does not provide that facility.

The converter is housed in a standard tin plate enclosure measuring 37mm x 74mm x 30mm and the thoughtful arrangement of the connections provides space for the RTL-SDR module.

The connection to the computer is via a USB cable that also provides the power supply. The small shortwave receiver can have the SDR PCB fitted inside the housing. The standard version has the USB stick connected outside the conver-

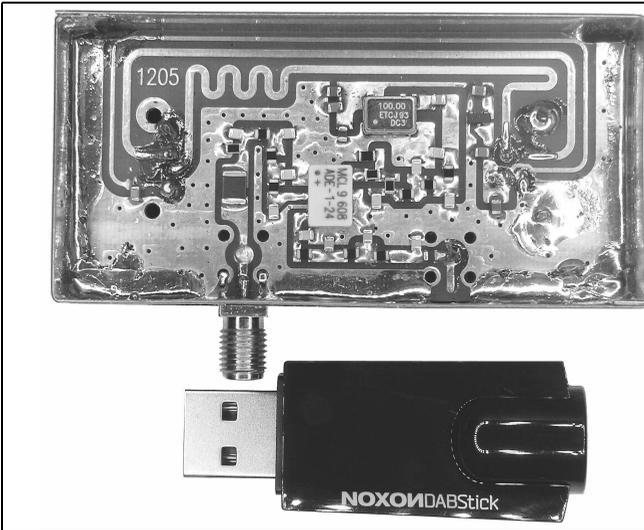


Fig 3: The compact shortwave converter developed specifically for DAB sticks. The range is 0 to 55MHz converted to 100 to 155MHz. Also in the picture is a NOXON DAB stick.

ter so that the correct version for each can be chosen.

Two possible variants with internal SDR receivers are available as shown in Fig 4. One has the E4000 Ezcrap Board fitted and the other has the NOXON FC0013 with TerraTec chipset.

The layout of the E4000 is somewhat unfortunate because there are multiple heat sources in the vicinity to the reference oscillator. Fig 4 shows a custom cooling solution to minimise the thermal drift to improved performance for narrow band use.

The currently NOXON stick [2] contains the E4000 tuner and can be integrated

excellently into the converter due to its small size. A little more than thumbnail sized circuit board remains after removing the plastic casing with just the USB connector and aerial socket (Fig 5).

The FC0013 stick has a better distribution of heat due to the better board layout. It can be used in the enclosed converter housing without additional cooling and is stable. The custom cooler of the E4000 sticks transmits heat through the case making it stable. Depending on the thermal design used by the stick manufacturer you must decide whether you want to use the extra cooling when installing very small boards.

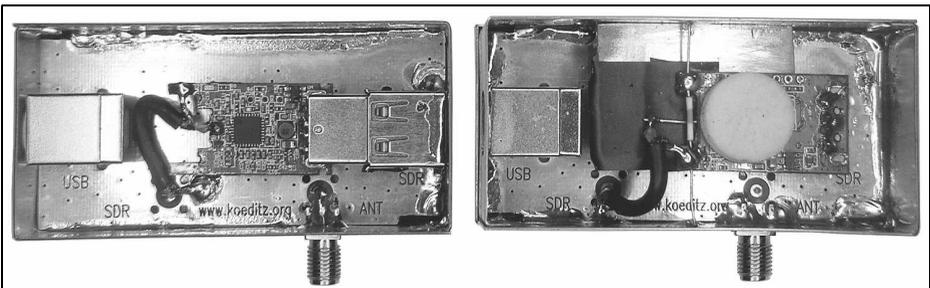


Fig 4: Two variants of the shortwave converter; On the right fitted with the E4000 Ezcrap PCB and on the left with a NOXON that uses the FC0013 chipset from the TerraTec.

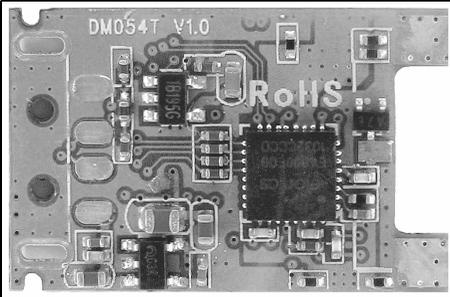


Fig 5: The naked NOXON stick prepared for installation in the SW converter.

The RTL SDR can be used in conjunction with this converter.

12.

Zadig operating multiple RTL-SDRs

Different versions of Zadig are available separately for Windows XP and other Microsoft operating systems. The current status of the software is zadig_v2.0.1.160 [3] that is available in two versions (for XP and Vista, 7 and 8).

The installation has not changed much,

all tuners run easily; using multiple drivers simultaneously still works with SDR#. Anyone who has tried this software already knows the small problems can be minimised by the correct choice of the boot order for the sticks, if not consult the first part [3]. Although much has improved, you will have to find the best way of using the software.

The offset frequency can be entered manually as shown in Fig 6 using the "shift" feature. The frequency displayed is then the actual frequency giving a real feeling that is a typical shortwave SDR.

This can be used with many sticks, just clone the SDR# program folder for each stick and rename the links. In this way simultaneous operation runs smoothly and different calibrations for each stick can be defined without constantly needing to re-adjust the tuner.

A powerful calculator was always required when using multiple tuners at the same time; in this case four instances of SDR# are being used. The combined operation of different tuners is now no longer a problem in the latest version of SDR#; the "configure" option allows the selective choice of various sticks correctly that are installed with Zadig (Fig 7).

The use of the "RF gain" works best with

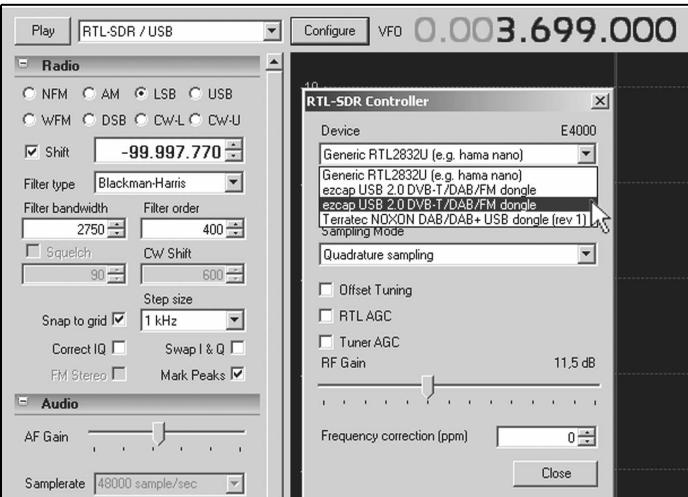


Fig 6: This shows 4 sticks configured in a system. There are two shortwave converters with FC0013 and E4000 devices and two regular sticks; E4000 and R820T.

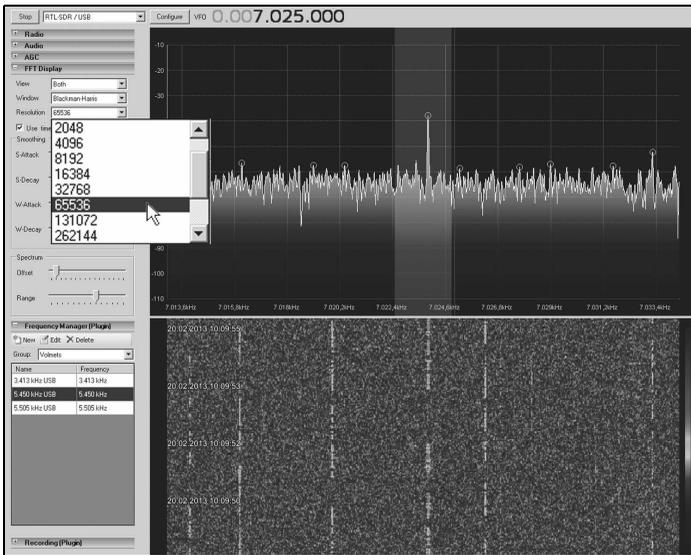


Fig 7: The spectral display uses alot of computer resources so it can lack resolution or fail in multi-mode operation when more resources are being used.

the E4000 tuner. Sometimes errors occur or the receiver seems deaf; if the tuner is correctly configured in the base configuration the SDR may be overloaded by strong input signals. Depending on the antenna and input level it is advisable to find the correct value of the gain so that the signals are noise free and an increase offers no audible benefits. This will also minimise the generation of mixing products. Depending on the tuner it is advisa-

ble to adjust the RF gain setting.

13.

HSDR and the RTL-SDR RTL2832U

Undoubtedly this program gives the sensation of a high end device as soon as it is

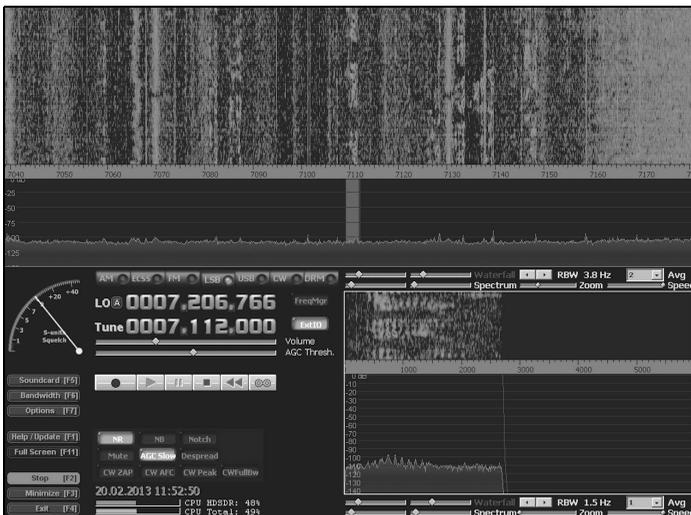


Fig 8: HSDR on the 40m band.

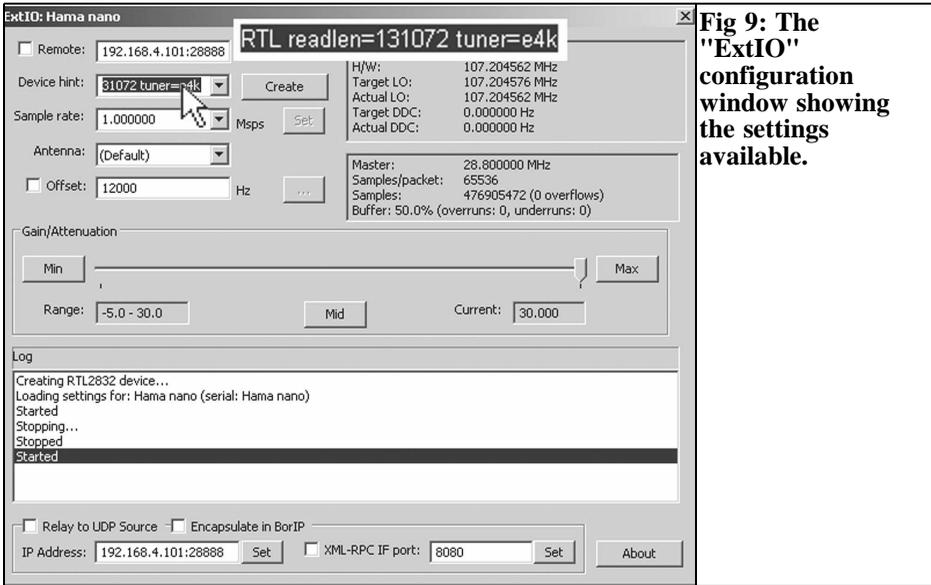


Fig 9: The "ExtIO" configuration window showing the settings available.

loaded. All these facilities at the ridiculous price of a re-used TV and radio stick (Fig 8). The Perseus amazed everyone with the facilities available and the software presentation. Unfortunately the cost of this device cannot be underestimated but you can come up with something similar with some meticulous work and free software.

In the first version of HSDR the RTL-SDR program squandered almost infinite resources. The developers have largely solved the problem and now HSDR uses a fraction of the resources used by the earlier version. The parameters "RBW" and "Speed" can be used to adjust the "Feel" and "Speed".

Interesting accessories such as memory banks and better integration of stick variants are missing but what can you expect from free software. You must be just grateful that it runs very well after some training and little expense.

Using the "borIP" tool supplied you can send the IQ stream across a network to realise an economical remote SDR server at a quieter location for reception.

13.1. HSDR settings for the RTL stick

Unfortunately all parts of the program needed to receive signals are not immediately available from the normal sources on the Internet.

The basic requirements are the same as for SDR#, but you need the "ExtIO" file and "Plug-ins" for the chipset of the tuner being used. There is a download folder [5] for my favourite the E4000 containing the additional parts of the program. Copy those files to the root folder of the fresh HSDR install and ideally you will only need to make the configurations shown in Fig 9:

Clicking on "ExtIO" in the main window and open the configuration menu and enter "RTL readlen = 131072 tuner = e4k" in the "device hint" followed by a click on the button "create". Manually choose the bandwidth and "sample rate" to get a waterfall display. The parameter "readlen" can be repeatedly multiplied or divided by two to find the best package size for your machine's hardware performance without generating artefacts.

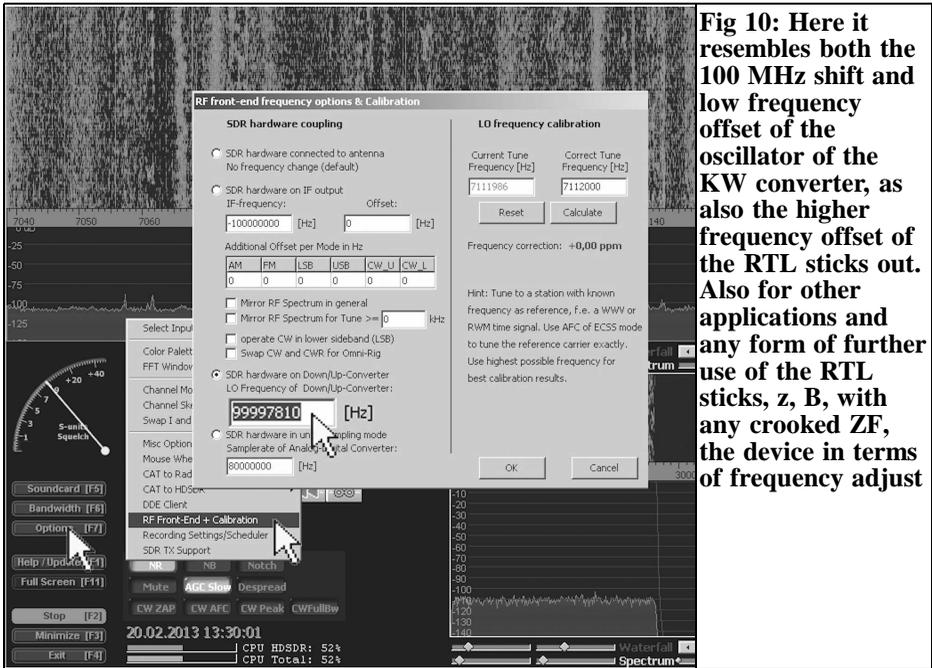


Fig 10: Here it resembles both the 100 MHz shift and low frequency offset of the oscillator of the KW converter, as also the higher frequency offset of the RTL sticks out. Also for other applications and any form of further use of the RTL sticks, z, B, with any crooked ZF, the device in terms of frequency adjust

Take out the frequency correction in the main window, click on "options" and continue in the submenu with "RF front end calibration".

If all went well you can now take full advantage of the other benefits of HSDSR compared to SDR#. This includes the bandwidth adjustment in the receive window, multiple configurable notch, a better approach to the squelch and more options for the output sampling rate for use in software decoders.

14.

Now for the measurement results

For all measurements an R&S CMT55 FM generator was used on 5kHz. The audio signal was taken from the output jack of the Sony laptop to a radio measuring station. The noise figure measurement runs automatically from the CMT and stops when the pre-set SINAD value is reached. The results are given in μV and dBm.

The E4000 tuner was measured first. The

Table 1: E4000 tuner using SDR#.

SINAD	144MHz	432MHz	1296MHz
12dB	0.284 μV -117.9dBm	0.228 μV -119.8dBm	0.309 μV -117.2dBm
20dB	0.452 μV -113.9dBm	0.281 μV -118.1dBm	0.486 μV -113.6dBm
30dB	1.060 μV -106.5dBm	1.190 μV -105.5dBm	1.600 μV -102.9dBm

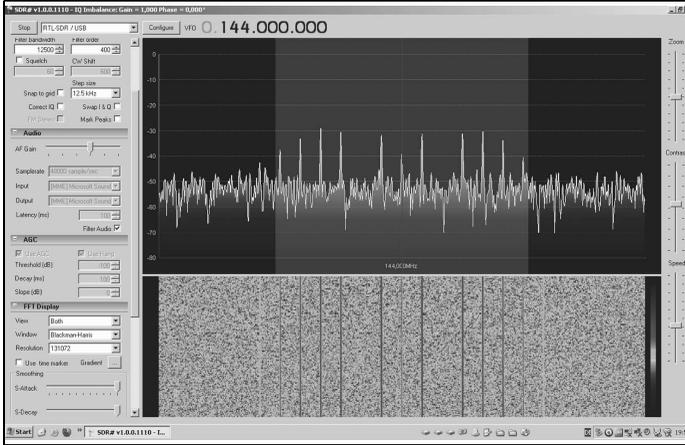


Fig 11: Signal quality at 20dB SINAD noise ratio (FM with 5kHz modulation).

RF gain of the tuner was set to maximum (+42dB) for the sensitivity measurement. The latest version of SDR# was used (RTL AGC on, tuner off, RF gain max.). The volume control was set to the central position so that no additional distortion was generated in the sound card (Table 1).

The next candidate was the R820T. All adjustments were made as above. This tuner was set to +49.2dB. Table 2 and Fig 11 show the results.

Due to the high sensitivity of the R820T tuner it was overloaded and a 2.4kHz SSB filter was used to obtain comparable

values. In order to determine the signal to noise the frequency was adjusted so that the carrier with a 1kHz tone was exactly 1kHz below the receiver display. This was completely unusual using the radio measuring station for FM signals. At 10dB s/n ratio is fabulous with a sensitivity of 0.063µV (Fig 12)!

The IP3 measurements in the next section show whether they are usable with an antenna (This refers to the maximum RF gain with full AGC that shows clipping of a -50dBm signal on the waterfall display from larger signals).

The E4000 was measured using the

Table 2: R820T using SDR#.

SINAD	144MHz		432MHz		1296MHz
12dB	0.162µV	-122.8dBm	0.145µV	-123.7dBm	0.404µV -114.9dBm
20dB	0.193µV	-121.3dBm	0.222µV	-120.1dBm	0.509µV -112.9dBm
30dB	0.608µV	-111.3dBm	0.672µV	-110.4dBm	1.600µV -102.9dBm

Table 3: E4000 using HSDR.

SINAD	144MHz		432MHz		1296MHz
12dB	0.241µV	-119.3dBm	0.382µV	-115.3dBm	0.406µV -113.7dBm
20dB	0.618µV	-111.2dBm	0.674µV	-110.4dBm	1.690µV -102.4dBm
30dB	1.710µV	-110.4dBm	1.170µV	-105.6dBm	2.490µV -99.1dBm

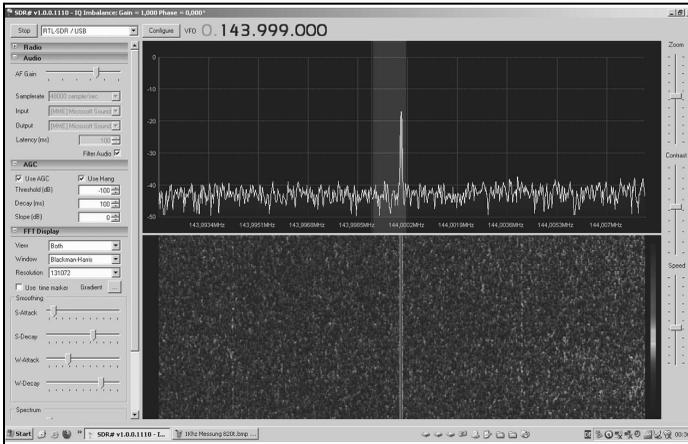


Fig 12: SSB measurement with an input signal of 0.603µV! The high signal to noise ratio on the FFT display should be noted; signals could even be detected with a -140dBm signal from the measuring equipment.

HDSDR program to compare the software. The software only processes the stream of data from the stick (Table 3).

As expected the results varied between the two programs because they use different algorithms.

14.1. IP3 measurements

In the VHF/UHF bands the sensitivity of the tuner is an advantage but in the shortwave range it is often harmful. Therefore, the measurement of large signal behaviour in the shortwave range follows. This used my own IP3 generator that produces an up to +55dBm signal for IP3 measurements (Fig 13). The output

level was set to 2 x 13dBm (Fig 14) corresponding to a strong evening signal on 40m from a freely suspended dipole. The measurements are only qualitative because the levels of the SDR programs have no referenced dB values. However the differences can be seen in direct comparison to identical conditions.

Measurements on the two SDR sticks using the FC0013 and E4000 tuners with the RF gain set to optimum for large signal behaviour showed strong distortion.

The E4000 is not best in class for sensitivity in the VHF/UHF range but the developer has taken more trouble with the RF control. For a DVB-T stick it makes a very good impression (Fig 15). With an additional input filter it can certainly match many commercial short wave transceivers if not even better results, especially with HDSDR that is the best program for shortwave.

The RF measurements were performed in the RF laboratory [1].

14.2. Conclusion of the previous measuring results

On-going improvements to the software and design changes to the RTL SDR indicate a trend towards better product quality. These measurement results are only valid for the components used in the



Fig 13: IP3 generator with a maximum output signal of 2 x +13dBm.

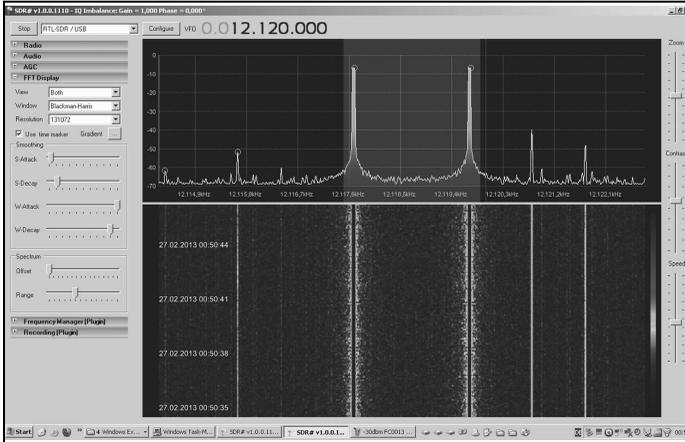


Fig 14: The 2 x 13dBm from the RF amplifier had no effect on the tuner. The tuner showed no sign of overload therefore it is suitable for use with a large antenna systems.

tests. The performance is unbeatable, especially for this low-cost category. Of course it would be nice if future generations of SDR sticks would use a higher bitrate so that the dynamic range could be improved. But they are already useable in the local shack or on the road they are handy as a holiday receiver. So you can indulge in shortwave radio with a few meters of wire.

RF man, wanted to know how the shortwave converter with an RTL-SDR stick performed in a real test. He successfully tested the converter in addition to the SDR at night on the 40m band. If you want to get an audible and visual impression of the shortwave converter with an SDR stick using the HDSDR program it can be found on the DK3WM homepage [6]. This is also suitable to verify your own transmitter signal including the modulation bandwidth.

15.

Shortwave reception test

Werner, DK3WM, who is an experienced

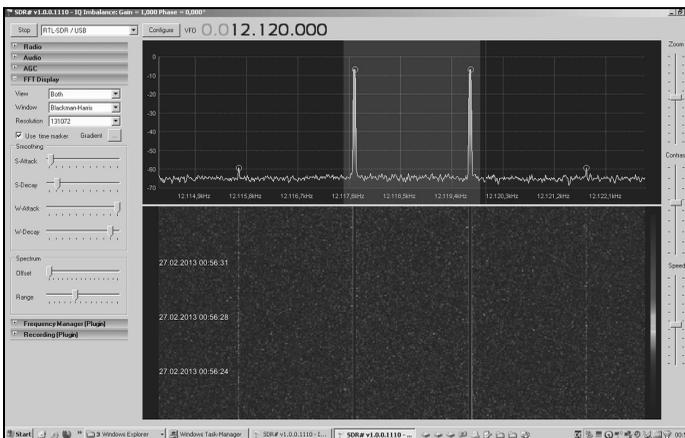


Fig 15: The E4000 tuner shows an almost perfect large signal behaviour.

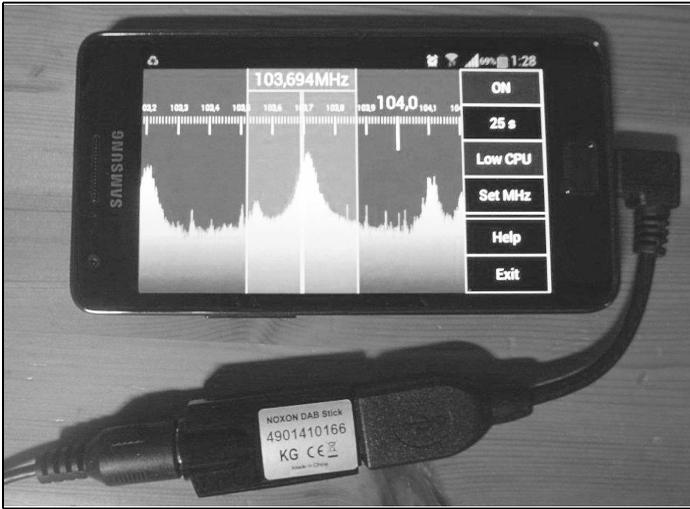


Fig 16: The software for Android shown on a Samsung Galaxy S2.

16.
SDR “on the go”

There is even the first program for a Smartphone to use as an SDR receiver [7] (Figs 16 and 17). Thanks to Klaus, DG1FFD for the information.

17.
References

[1] Reference of the shortwave Converter: Frank Köditz telecommunications;

- 1205 Model: www.koeditz.org
- [2] Reference for the Noxon sticks from Conrad Electronic: www.conrad.com
- [3] Driver: [zadig_v2. 0.1.160](http://sourceforge.NET/projects/libwdi/files/zadig/zadig_v2.0.1.160.7z/download)
- http://sourceforge.NET/projects/libwdi/files/zadig/zadig_v2.0.1.160.7z/download
- [4] The RTL-SDR, Working with a USB-stick, Dirk Müller, DB6FM, VHF Communications Magazine issue 2/2013 pp. 82 - 89
- [5] Download folder: www.vhfcomm.co.uk/download/SDR-3-2013
- [6] Video stream from DK3WM: <http://www.OV-f73.de/Technik/SDR-1/SDR-experiment/>
- [7] <http://forum.xda-developers.com/showthread.php?p=36930621>



Fig 17: The software is called SDR-Touch.



Harald Braubach, DL1GBH

Directional couplers - made to measure

Reprint from issue 4/1984

1.

Introduction

It was always difficult for radio amateurs to construct wideband directional couplers having a low coupling attenuation. Microstrip couplers are easy to manufacture for those that have such capabilities. However, the minimum coupling attenuation that can be obtained with a reasonable directional characteristic is in the order of 10dB. On the other hand, it is virtually impossible, using microstrip technology, to design 3dB power dividers, such as are required when constructing push-pull mixers, or for feeding circular polarised antennas. It is possible, of course, when using tri-plate circuits for these values to be achieved, however, the conductor tracks are then so thin that it is hardly possible to use them in conjunction with higher power levels. Most radio amateurs do not have the

necessary machinery to construct conventional directional couplers mechanically, and do not have enough room for accommodating such large couplers.

A good solution for solving the problem of home made directional couplers is offered by a product manufactured by Sage Laboratories Inc. called "Wireline" and "Wirepac". It is possible using both these systems to construct directional couplers in the range of 3 to 20dB coupling attenuation in a frequency range from 50MHz to 2.4GHz. Wireline is the cheaper of the two and has a directivity of 20dB. Wirepac has a directivity of 30dB, but is considerably more expensive, and is therefore not to be discussed here.

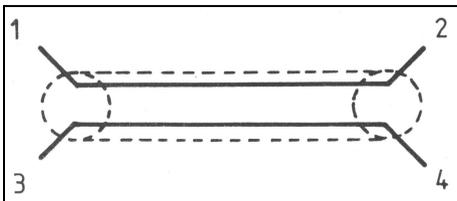
2.

Fundamentals

The Wireline type to be described is a line directional coupler and comprises two coupled lines as shown in Fig 1. The coupling attenuation is dependent on frequency and achieves its minimum value at a coupling length of $\lambda/4$ (see Fig 2).

Under matched conditions (Fig 3), the following is valid:

If a signal with a power P_1 is fed to the

**Fig 1:**

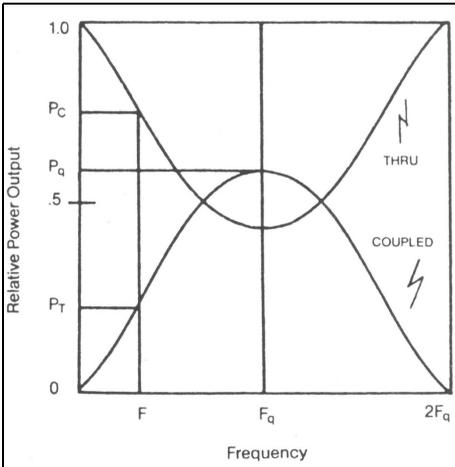


Fig 2: Coupling attenuation and insertion loss as a function of frequency.

input, a power of $P_2 = P_1 - P_1 \times c$ will be present at R_2 , and a power of $P_3 = P_1 \times c$ at R_3 , where $c =$ coupling factor.

In the case of an ideal directional coupler, R_4 will be powerless, since the diagonally opposite inputs are decoupled from one another. In practice, a power will be present that is reduced to the value of the directivity d .

$$P_4 = P_1 \times c \times d \quad (d = \text{directivity})$$

accordingly

$$P_3 = P_1 \times c - P_1 \times c \times d$$

A further characteristic of directional

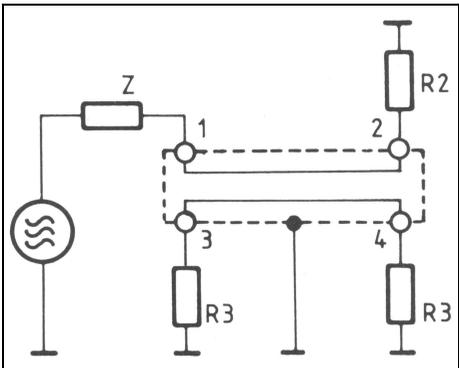


Fig 3: Matched directional coupler.

couplers is that the signals of the coupled outputs will have a frequency independent phase difference of 90° .

3.

Construction of wireline

There are five different versions that differ in the type of screening and the maximum power ratings. The internal construction is shown in Fig 4. The following Table 1 shows the most important differences between the individual types.

Due to the coaxial type construction of the coupler, it is possible for the two

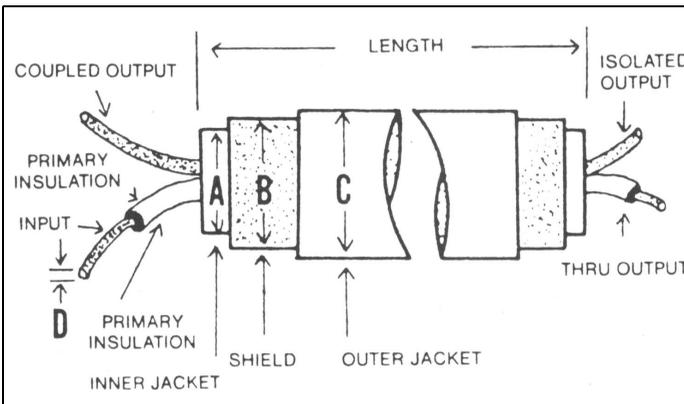


Fig 4: Construction of Wireline.



Table 1: Wireline Designs. P_m = mean power, P_p = peak value of the power rating.

Type	H	HB	HC	JB	JC
Shield	Double foil screen	Copper mesh	Copper tube	Copper mesh	Copper tube
P_m/W	100	100	100	200	200
P_p/W	2000	2000	2000	2000	2000

coupled outputs to be provided on one side as shown in Fig 5. This offers several advantages for practical construction.

4.

Calculation of the couplers

4.1 Calculation of a Coupler with a certain Coupling Attenuation at a certain Operating Frequency

The following data is required for the calculation:

- Required centre frequency f_{op} (e.g. 435MHz)
- Required coupling attenuation a_c (e.g. 10dB).

It is firstly necessary to convert the logarithmic value of the coupling attenuation a_c into the linear coupling factor c .

$$c = 10^{-\frac{a_c}{10}} \tag{1}$$

In the case of a 10dB coupler, the following results:

$$c_{10dB} = 10^{-\frac{10}{10}} = 10^{-1} = 0.1$$

Now the frequency at which 3dB coupling is achieved is calculated, using the operating frequency f_{op} and the coupling factor of the frequency f_c .

$$f_c = \frac{90f_{op}}{\arcsin \sqrt{-\left(\frac{c}{c-1}\right)}} \tag{2}$$

The following will result using the values of $f_{op} = 435MHz$ and $c = 0.1$

$$f_{c(10dB/435)} = \frac{90 \cdot 435MHz}{\arcsin \sqrt{-\left(\frac{0.1}{0.1-1}\right)}} = 2010.66MHz$$

From this quarter wave frequency (f_c) one then calculates the length l of the coupler as follows:

$$l = \frac{4700MHz \cdot cm}{f_c(MHz)} \tag{3}$$

This results in the following coupler length in our example:

$$l_{(10dB/435)} = \frac{4700cm}{2010.66} = 2.338cm$$

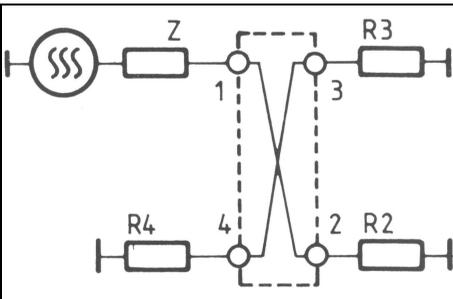


Fig 5: Coupler showing the coupled outputs on one side.



Table 2: Directional coupler, length l = 50mm, coupling attenuation as a function of frequency.

f MHz	a _c dB
3.5	44.66
7.0	38.64
14.0	32.62
21.0	29.10
28.0	26.61
145.0	12.64
435.0	5.13

A 10dB coupler at 435MHz would therefore have a length of 23.38mm.

4.2 Calculation of the Coupling Attenuation of any required Coupler

The following data is required for calculation:

- Length (l) of the coupler in cm (e.g. 10cm)
- Frequency (f) at which the coupling attenuation is to be calculated (e.g. 435MHz)

Firstly find the quarter wave frequency (f_c) of the coupler:

$$f_c = \frac{4700MHz \cdot cm}{l(cm)} \quad (4)$$

In our example:

$$f_{c(10cm)} = \frac{4700MHz}{10} = 470MHz$$

This is followed by calculating the coupling factor (c):

$$c = \frac{\sin^2\left(90 \frac{f}{f_c}\right)}{\sin^2\left(90 \frac{f}{f_c}\right) + 1} \quad (5)$$

In our example:

$$c_{(10cm435)} = \frac{\sin^2\left(90 \frac{435}{470}\right)}{\sin^2\left(90 \frac{435}{470}\right) + 1} = 0.4966$$

The coupling attenuation (a_c) is now calculated from the coupling factor:

$$a_c = -10 \log c \quad (6)$$

The following will result in our example:

$$a_{c(10cm435)} = -10 \log 0.4966 = -3.04dB$$

5.

Practical applications of Wireline

5.1 Use as a Directional Coupler

Of course, the primary use of Wireline couplers is for determining the VSWR of antennas and other loads. The construction of VSWR bridges is not to be discussed, since it is well known. Table 2, however, provides an aid for designing a directional coupler for frequencies up to 435MHz.

5.2 Use as a 3dB Coupler

This results in a multitude of applications of which the most important are to be mentioned.

5.2.1 Feeding of Circular Polarised Antennas

Since the coupled outputs always possess a phase shift of 90° (± 1°) to one another, it is easily possible to construct a low loss, wideband feed for circular polarised antennas (see Fig 6).

Directional couplers as shown in Figure 5 behave as a 4/4λ hybrid (see Fig 7). An RF voltage fed to 1, or A, will be distributed equally to 2 and 3, or C and

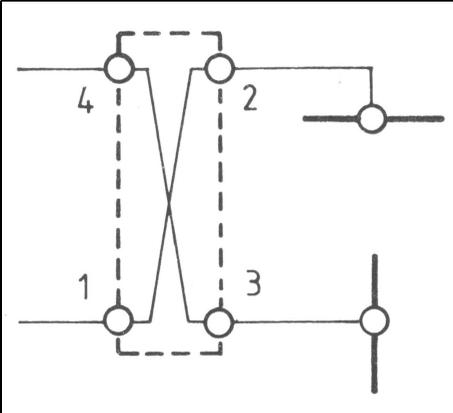


Fig 6: Directional coupler for feeding circular polarised antennas.

D. Connection 4, or B, remains decoupled. An RF voltage fed to 4, or B, will be distributed equally to 2 and 3, or C and D. In this case, 1, or A will remain decoupled.

Connections 2 and 3, or C and D, have a phase shift of 90° to one another (this will only be the case at the centre frequency of a $4/4\lambda$ hybrid).

If, for instance, an RF signal is fed to 1, and 4 is terminated with 50Ω , anticlockwise, circular polarisation will result. If, on the other hand, 4 is fed with the RF voltage, and 1 is terminated with 50Ω , clockwise, circular polarisation will result. Of course, the actual polarisation will also be determined by the phase position of the individual antenna. An anticlockwise circular polarisation will

be changed to clockwise polarisation on rotating the phase position of one of the antennas by 180° .

As can be seen, the polarisation switching is nowhere near as critical as when using conventional coaxial delay line methods, and where the switching relay must be taken into consideration in the phase shift calculation. In the case of the described type of feeding, the relay is placed in front of the phase shift 3dB coupler (Fig 8). Attention must only be paid that the lengths of the antenna feeders are identical. The terminating resistors should have a rating of one 100th of the transmit power if the antenna matching is good.

5.2.2 Construction of Push-Pull Mixers

A further application of Wireline 3dB couplers is given in the construction of push-pull mixers (Fig 9). A mixer constructed in this manner will have a bandwidth of one octave (frequency ratio 1:2).

5.2.3 Construction of Wideband Power Amplifiers

At higher frequencies, it is difficult to connect wideband amplifiers in parallel to achieve higher power levels. In most cases, $4/4\lambda$ hybrids are used. This means that it is possible to use Wireline 3dB couplers here, which also have the advantage of being much smaller (Fig 10).

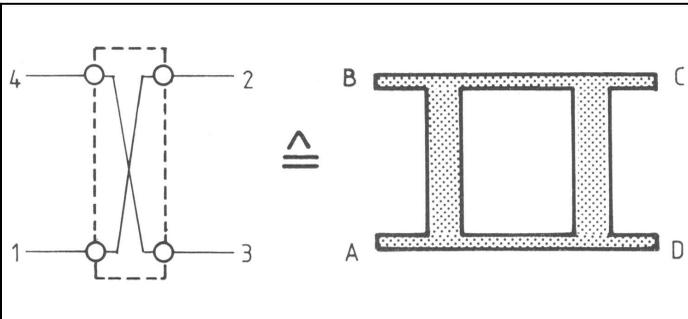


Fig 7: Comparison between a wireline coupler and a $4/4\lambda$ hybrid

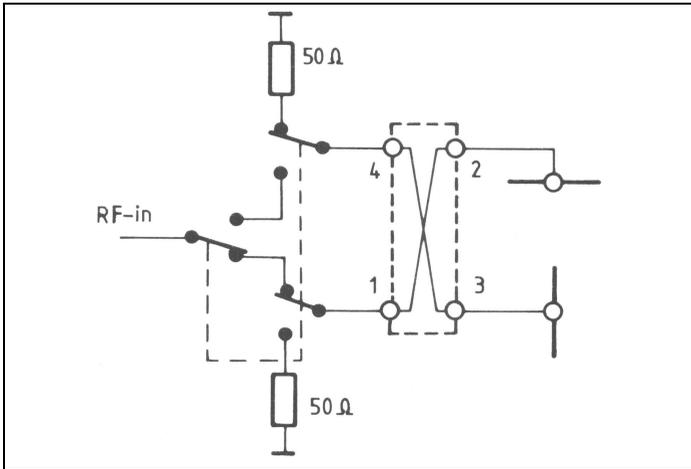


Fig 8: Circuit for switching the polarisation of circular antennas.

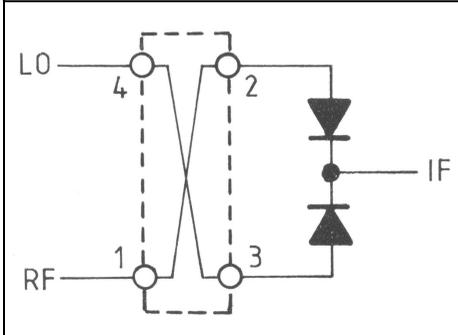


Fig 9: Push-Pull mixer with 3dB coupler.

6.

Manufacturer availability of Wireline and design programs

Wireline is available from Sage Laboratories Inc. or from Wacker at the address given below.

6.1 Program

A German company has a basic program for the TRS-80 M III for the design of such couplers. We would like to suggest that interested readers contact this company directly. The address is as follows:

Firma Wacker GmbH, Grunenburgweg 85, D 6000 Frankfurt 1/West Germany.

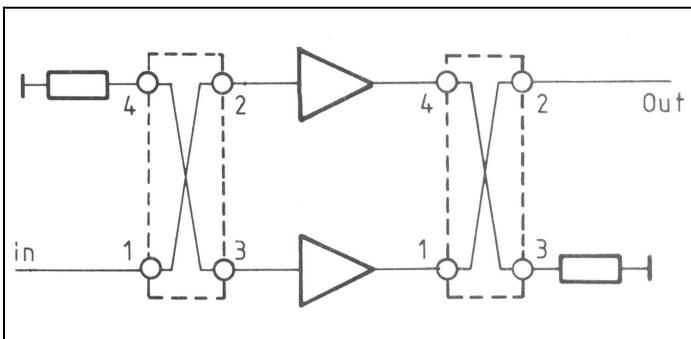


Fig 10: Wideband amplifier with directional coupler.



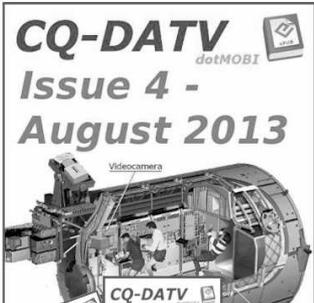
f MHz	3dB	6dB	10dB	20dB	Table 3: Coupling length as a function of coupling attenuation and frequency.
3.5	too long	too long	too long	860mm	
7.0	too long	too long	too long	430mm	
14.0	too long	too long	726mm	215mm	
21.0	too long	880mm	484mm	143mm	
28.0	too long	660mm	363mm	107.5mm	
145.0	324mm	127.5mm	70.1mm	20.7mm	
435.0	108mm	42.5mm	23.4mm	too short	
1275.0	36.9mm	too short	too short	too short	
2350.0	20.0mm	too short	too short	too short	

Note: Since the article was first published some details have changed. Information about Wireline can be found at:

http://micro.apitech.com/wireline_wirepac.aspx

and a calculation program from:

<http://www.rfcafe.com/business/software/wireline/wireline-calc.htm>



CQ-DATV
dotMOBI
Issue 4 - August 2013

Welcome to CQ-DATV Magazine
CQ-DATV is free eBook format publication focused on ATV, but lots of the constructional projects have Amateur Radio applications. There is no membership, logins, or email required. Issues 1 to 5 available now

www.cq-datv.mobi





CQ-DATV
Issue 6 dot.MOBI
Out November 2013





The UK Six Metre Group, www.uksmg.com

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

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



Michael Martin, DJ7VY

Extremely Low Noise Preamplifiers require Low Loss Antenna Cables! Wideband Directional Coupler for VSWR Measurements on Receiver Systems

Reprint from 3/1983

The use of GaAs-FET technology has brought a considerable increase in receiver sensitivity that was thought impossible several years ago. Both bipolar transistors and GaAs-FETs now offer noise figures of less than 1dB, and the latter even allow noise figures of less than 0.5dB to be achieved. This means that system temperatures of 50K = -223°C are now possible using this technology, which were only possible with cooled parametric amplifiers in the past. The following article is to discuss several special features of the GaAs-FET preamplifiers without which it is not possible to obtain the values given in the data sheets. Furthermore, a wideband directional coupler is to be described that allows VSWR measurements to be made on the input circuits of receivers and preamplifiers in the frequency range of 2 to 1400MHz.

1. General

A typical receive system comprises a receive antenna a short piece of feeder cable between antenna and masthead preamplifier a second longer cable to the actual receiver. The task of the antenna is

to receive as much energy as possible. This energy should be amplified without any deterioration up to the demodulation level; this is obtained using special low-noise preamplifiers at RF level.

2. Noise

2.1. Thermal Noise

Thermal molecule movement causes noise which tends to blanket very low receive signal levels. The natural limit value is given by the noise of the input impedance of the receiver which amounts to 290K at an ambient temperature of 17°C. This corresponds to a noise power of -174dBm per Hz of bandwidth. In the case of a conventional bandwidth of 2.4kHz, this is equal to an input noise power of -140dBm, corresponding to an input voltage of 22nV into 50Ω. This means that no signals of less than 22nV can be received, even when using an ideal noiseless receiver that is connected using a lossless cable to the antenna, if the bandwidth is not to be decreased.

2.2. Signal-to-Noise Ratio (Noise Figure NF)

The noise figure is used to define the

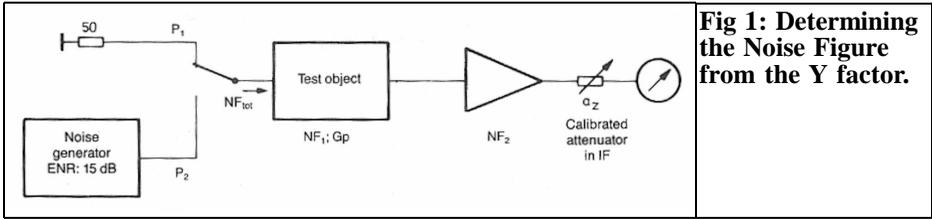


Fig 1: Determining the Noise Figure from the Y factor.

quality of an amplifier. This definition indicates how much the signal-to-noise ratio at the output of the amplifier has deteriorated with respect to that at the input.

The noise factor F is always >1 and is obtained according to the following equation:

$$F = \frac{S_{in}/N_{in}}{S_{out}/N_{out}} \quad (1)$$

The noise figure is given in dB and is obtained from the noise factor with the aid of logarithms:

$$NF_{dB} = 10 \lg F \quad (2)$$

An ideal amplifier would have a noise figure of $F = 1 \equiv NF = 0\text{dB}$, and the input signal-to-noise ratio would be present at the output without change. A true amplifier with a noise figure of 3dB will reduce a signal-to-noise ratio of 10dB coming from the antenna to a value of $10 - 3 = 7\text{dB}$ at the output. The target of all amplifier developments is to obtain a noise figure that is less than 1dB. Improvements in the order of 0.5dB can result in considerable system improvements in the case of EME communications where the antenna is pointed towards cold space [1]. It is possible to achieve noise figures of 0.5dB using the present state-of-the-art on all amateur bands between 145 and 1296MHz.

2.3. Cable Noise

Cables, attenuators, input resonant circuits, and all other passive four-poles will cause noise to the value of their insertion

loss: $NF = a_0(\text{dB})$. A cavity filter having an insertion loss of 0.1dB will deteriorate the input noise figure by the same value.

2.4. Contribution of the Second Amplifier Stage

The second amplifier stage of our receiver system influences the overall noise figure according to equation 3:

$$F_{tot} = F_1 + \frac{F_2 - 1}{G_1} + \dots + \frac{F_n - 1}{G_n} \quad (3)$$

It will be seen that the higher the preamplification, the less will be the contribution of the second stage to the overall noise figure F_{tot} .

2.5. Measuring the Noise Figure

The most favourable alignment of preamplifiers with a continuous noise figure measurement is possible with equipment operating according to the PANFI principle (Precision Automatic Noise Figure Indicator) by which the Y factor of the amplifier is re-calculated into noise figure and is continuously indicated [2]. The measuring system for determining the noise figure with the aid of the Y factor is shown in Fig 1.

According to equation 4 the following results:

$$NF_{dB} = ENR_{dB} - 10 \lg(Y - 1) \quad (4)$$

where

$$Y = \frac{P_2}{P_1} = 10 \frac{a_z}{10^3}$$

The value a_z should be measured with an

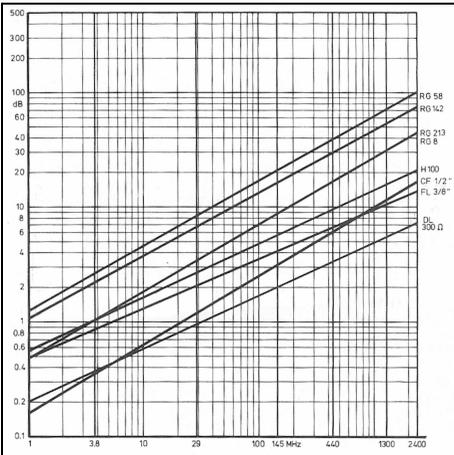


Fig 2: Attenuation a_0 of various coaxial cables, as well as a 300Ω twin line shown as a function of frequency for a cable length of 100m.

3. Characteristics of Antenna Cables

All cables between the antenna and the preamplifier possess an insertion loss that will deteriorate the system noise figure by at least the value of its insertion loss. This is not only a function of the cable length, but is also dependent on the impedance ($Z_{opt} = 75\Omega$ for CATV), and the terminating resistance. The insertion loss is a minimum when the impedance of the antenna corresponds to the Impedance Z_0 of the cable, and this to the input impedance Z_{in} of the preamplifier.

If there is a difference between Z_0 and Z_{in} standing waves will be generated in the cable that cause ($I^2 \times R$) losses at the points of maximum current, and lead to greater dielectric losses at the points of maximum voltage. Cables having gas or PTFE insulation using thick, polished conductors will have lowest losses. Fig 2 shows a diagram in which the loss of the most common cables is given for a length of 100m as a function of frequency. The difference slope of the lines shows the different distribution between copper and dielectric losses.

In the ideal, matched case, the following will result, for instance, with two metres of RG213 cable having $a_0 = 0.32\text{dB}$ at 435MHz in front of a preamplifier with a NF of 1dB and an input impedance of 50Ω and a VSWR of 1.0: Overall noise figure $NF_{tot} = 1.0 + 0.32 = 1.32\text{dB}$.

accuracy of $\pm 0.05\text{dB}$.

ENR = Excess Noise Ratio = Noise power of the noise generator in dB; Y factor: Ratio of the output noise power of the amplifier with the noise generator switched on and off.

If an additional attenuation of, for instance, $a_z = 14\text{dB}$ must be inserted after switching, in order to obtain the same meter reading with a noise generator having an ENR of 15dB, the following will result from equation 4:

$$NF = 15 - 10 \lg(10^{1.4} - 1) = 1.17\text{dB}$$

An a_z value of 14.5dB, on the other hand, would result in an NF value of 0.65dB. It will be seen from equation 4 that any inaccuracy of the ENR calibration of the noise source will have an immediate effect on the test result. A further difficulty in the case of absolute measurements is that only very good calibration attenuators allow a reproducible accuracy of better than 0.1dB, which is especially required when low noise figures are to be measured.

4. Characteristics of Preamplifiers

Both in the case of preamplifiers equipped with bipolar transistors and those equipped with FETs, there is a physically dependent difference between their input

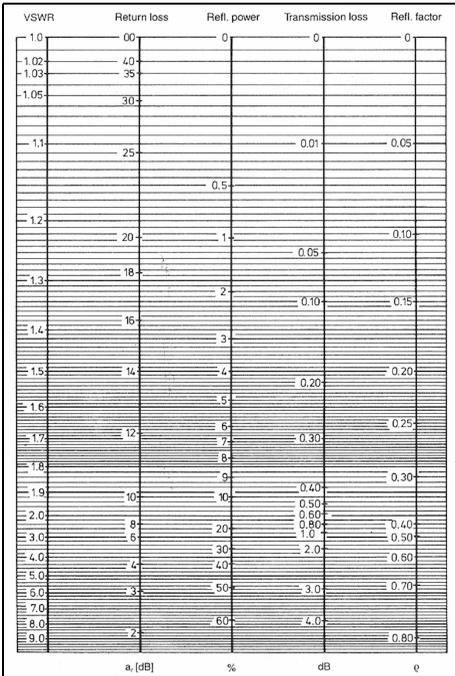


Fig 3: Relationship between return loss a_r , VSWR and reflection factor ρ .

impedance and the generator impedance required for obtaining a minimum noise figure. This is the well known difference between power and noise matching. It was possible in [3] to obtain an approximation of both values, however, it was necessary to reduce the gain per stage to approximately 10dB, which resulted in the contribution of the second stage to amount to one ninth of the total value (see equation 3).

Several preamplifiers were examined with the aid of a directional coupler bridge with respect to their input matching. The following relationship exists between VSWR and return loss according to equation 5:

$$VSWR = \frac{1 + 10^{-0.05a_r}}{1 - 10^{-0.05a_r}} = \frac{1 + \rho}{1 - \rho^3}$$

ρ = reflection factor

This can be determined graphically using Fig 3.

Bipolar amplifiers with feedback, exhibited values in the order of $a_r = 12\text{dB} \equiv VSWR = 1.67$ with a NF of 1.2dB, whereas GaAs-FET amplifiers with a NF of 0.6dB exhibited a_r values of only 2dB $\equiv VSWR = 8.72$. In the case of special wideband types VSWR values of between 17 and several hundred were exhibited. Especially low noise circuits with noise figures of less than 0.5dB exhibited, sometimes, negative input impedances.

It was found in many measurements of the cable losses in a measuring system comprising noise generator - cable to be measured - preamplifier - PANFI that the determined values differed greatly from the theoretical values ($NF_{tot} = NF_{prea} + a_0$). This is caused by the fact that residual reactive components of the generator impedance and the resulting impedance transformation in the cable can falsify the noise matching of the amplifier. These values will differ even more, the higher the input VSWR of the preamplifier, and the longer the interconnection cable.

5.

Examples

A few examples are now to be given to show the effect of attenuation on the system noise figure.

5.1.1. Preamplifier equipped with bipolar transistor:

$NF = 1.2\text{dB}$, $a_r = 12\text{dB}$, $VSWR = 1.67$
 When using a 10m length of RG213 cable at 435MHz the following will result from Fig 2:

$$a_0 = 1.6\text{dB};$$

$$NF_{tot} = 1.6 + 1.2 = 2.8\text{dB}$$



5.1.2. Same amplifier located close to the antenna:

$$2\text{m RG213, } a_0 = 0.32\text{dB,}$$

$$NF_{\text{tot}} = 1.2 + 0.32 = 1.52\text{dB}$$

NOTE:

Low Noise Preamplifiers should always be mounted in the vicinity of the antenna!

5.2.1 Preamplifier with GaAs-FET:

$$NF = 0.6\text{dB}$$

$$10\text{m RG213: } a_0 = 1.6\text{dB}$$

$$NF_{\text{tot}} = 0.6 + 1.6 = 2.2\text{dB; what a waste of a GaAs-FET}$$

5.2.2 Same amplifier mounted in the vicinity of the antenna:

$$2\text{m RG213: } a_0 = 0.32\text{dB}$$

$$NF_{\text{tot}} = 0.6 + 0.32 = 0.92\text{dB}$$

5.3 Low Loss antenna cable

3/8" Flexwell cable with inner dia = 4.2mm, outer dia = 16mm: 0.13dB/2m at 435MHz.

5.3.1 Bipolar Amplifier

$$NF = 1.2\text{dB and VSWR} = 1.67$$

$$NF_{\text{tot}} = 1.2 + 0.13 = 1.33\text{dB}$$

5.3.2 GaAs-FET

$$NF = 0.6\text{dB and VSWR} = 8.72$$

$$NF_{\text{tot}} = 0.6 + 0.13 = 0.73\text{dB}$$

6.

Improving the Noise Figure by using Extremely Low Loss Feeder Cable

Twin line "DL" is constructed from 2.5mm dia enamelled copper wire with PTFE spacers at a spacing of $\lambda/2$, wire spacing 12.5mm, $Z_o = 298\Omega$, Velocity Factor VF = 0.95

Attenuation at 435MHz: 3.3dB/100m; 2dB/100m at 145MHz.

6.1 Bipolar Amplifier with 2m Twin Line

$$a_0 = 0.07\text{dB}$$

$$NF_{\text{tot}} = 1.2 + 0.07 = 1.27\text{dB}$$

6.2. GaAs-FET Preamplifier with 2 m Twin Line

$$a_0 = 0.07\text{dB}$$

$$NF_{\text{tot}} = 0.6 + 0.07 = 0.67\text{dB, which corresponds to a very low NF deterioration!}$$

RESULT:

The high impedance, low loss twin line is very suited to the high impedance input of the GaAs-FET preamplifier!

$$NF = 0.6\text{dB } 43\text{K noise temperature, } NF = 0.676 \equiv 49\text{K.}$$

NOTE:

In order to keep the effects of differing cable length in the system noise figure as low as possible, the intermediate cable should be as short and as low loss as possible. An exact solution is only possible by aligning the preamplifier in conjunction with the antenna and interconnecting cable by injecting a keyed noise power, using a second antenna, into the receive system and aligning it for minimum noise figure on the PANFI.

Since this is very extensive, and is usually not possible at most amateurs' locations, it can usually only be carried out in the "laboratory":

The generator impedance which the antenna offers to the preamplifier at the end of the feeder cable must be measured and the amount and phase must be simulated with the aid of a stub tuner between noise generator and preamplifier during the alignment for minimum noise figure.

A minimum noise figure alignment made in the laboratory in conjunction with a noise generator impedance of $50\Omega \pm 0.5\Omega \equiv a_1$ 40dB is only reproducible in practice

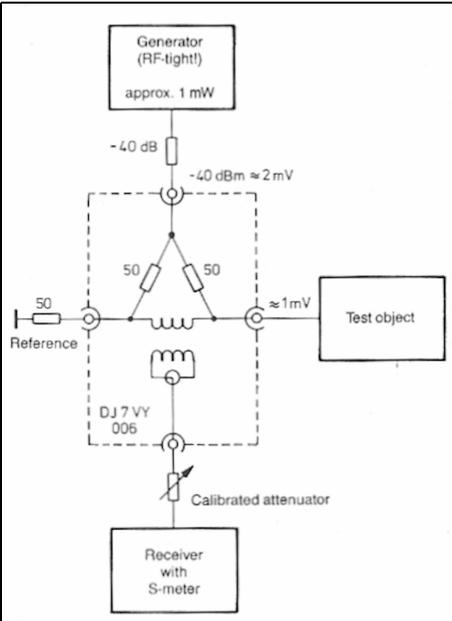


Fig 4: Measuring system for determining the return loss or preamplifiers.

if the antenna exhibits the same impedance!

Amateur antennas, on the other hand, some times have ar values between 10 and 20dB, which means that the results remain somewhat uncertain

7.

Measurement of Input Matching with the aid of a Directional Coupler

Since preamplifier and receiver input circuits are usually overloaded when using a power of 0dBm = 1mW, it is not possible to use conventional VSWR meters due to their insensitivity.

With the aid of the measuring system shown in Fig 4, it is possible for the ar value to be determined reliably at levels

of less than -40dBm.

A generator feeds a level of approximately -40dBm at the required measuring frequency to the bridge circuit. The power present at the input of the test object corresponds to the injected power minus 6dB. A 50Ω terminating resistor with the best possible matching is connected to the reference port, to which the test object is compared. The receiver is now connected via a calibrated attenuator with the output of the bridge. After disconnecting the test object, the receiver is adjusted to a certain S meter reading corresponding to approximately 60dB with the aid of the calibrated attenuator. If an identical terminating resistor is now connected to the test object port, the reduction of the S meter reading corresponds to the directional response of the bridge. This should be greater than 30dB, if possible, when low VSWR values are to be measured. After connecting the test object, the reduction of the S meter reading is directly proportional to the ar value in dB, if the S meter is also calibrated in dB. With the aid of the calibrated attenuator, a substitution measurement can be carried out by measuring this attenuation and reading it off on the attenuator.

It is then possible with the aid of equation 5 or Fig 3 to determine the VSWR. In order to be able to establish the overall system noise figure when using differing preamplifiers and cables, it is necessary for the input reflection values to be obtained from the manufacturer of the preamplifiers. A detailed description of further measurements that can be carried out with the aid of the bridge are to be found in [4].

8.

Construction

The directional coupler bridge can be

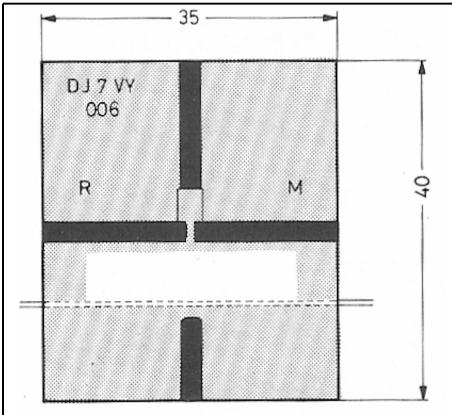


Fig 5: The double coated directional coupler board DJ7VY 006 is made from 1.5mm thick epoxy glassfibre material G10.

accommodated on the double coated PC board DJ7VY 006 and enclosed, as shown in Fig 5, in a metal box having the dimensions 35mm x 110mm. The main difficulty of this design is to be seen in the construction of the extremely wide band balun transformer which is similar to a description published by HP in [5].

The case is firstly provided with the four connectors, after their centre conductors have been shortened down to approximately 2mm. After cutting the board at the dashed line, cut-outs should be provided on the PC board to fit the protruding parts of the connectors and for the resistors. Caution should be paid that not too much material is removed! The PC board should fit in as well as possible into the top and bottom of the case

The two parts of the board are now soldered into place as shown in Fig 6 by firstly soldering the stripline connections after which the ground surface is soldered. The four 1% resistors that form the input resistors are now soldered into place at the location of the cut-out, and an approximately 90mm length of SM 50 PTFE cable is soldered to the output stripline. Place as many ferrite beads onto the cable until approximately 7mm spacing remains to the balanced centre

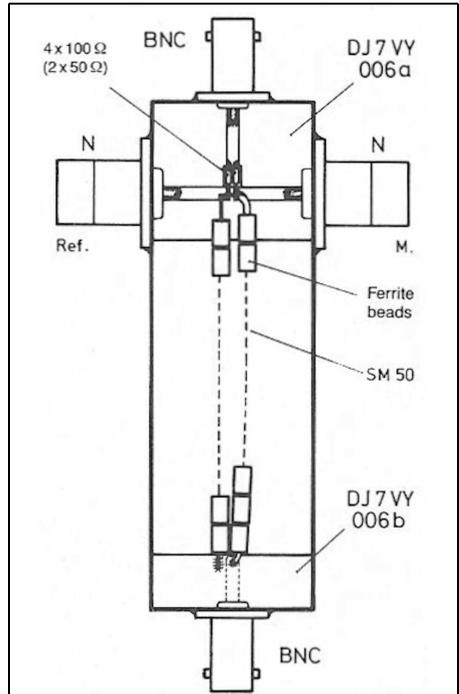


Fig 6: Partial PC board DJ7VY 006b must be rotated by 180° and soldered into place.

point. The inner conductor of the cable is connected to the reference stripline, and the outer conductor is connected to the measuring port stripline. A copper wire of 0.8mm diameter is also provided with ferrite beads and is soldered into place between ground and the reference line. The ferrite beads should be glued as shown in the prototype, using a normal adhesive so that the last 7mm remaining free. Important: The balun should run as horizontal as possible to the balanced point of the bridge! This is shown more clearly than in the text in the photographs of the author's prototype. Figs 7 and 8 show that two pieces of 51Ω resistors have been installed instead of the four pieces of 100Ω.

With a clean, balanced construction, the directivity of the bridge will be greater than 30dB! The values of the author's

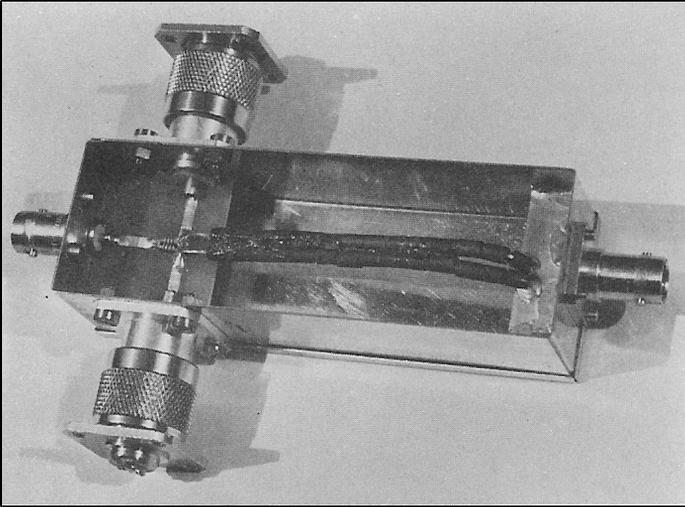


Fig 7: Photograph of the author's prototype seen as Fig 6.

prototype are given in Fig 9. It is possible to construct one's own terminating resistors instead of the very expensive precision terminating resistors described (see Fig 10). Two 1%-resistors of 100Ω , each, can be soldered onto a flange connector after filing down the centre pin to the same height as the outer collar. These terminating resistors exhibited a return loss in the order of 30dB even at 1296MHz. This is greatly suitable for amateur measurements, since the test objects to be measured will never be better than $1.2 \approx a_r = 20\text{dB}$.

Since the price of such homemade terminating resistors is low, it is recommended that identical 60Ω ($2 \times 120\Omega/1\%$) and also 75Ω ($2 \times 150\Omega/1\%$) types are constructed so that it is possible to measure VSWR values at other impedances.

CAUTION:

The input coupling resistors will not handle more than 0.1W. In the case of the author's prototype the lowest directivity values amounted to:

2MHz: -22dB, 4MHz: -30dB, 10MHz: -

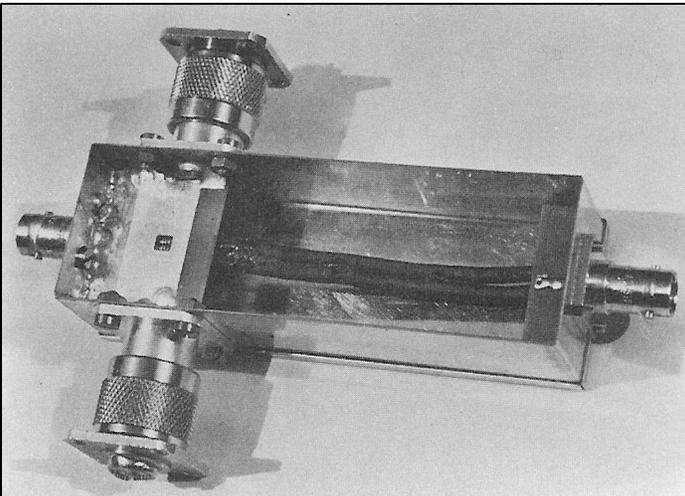
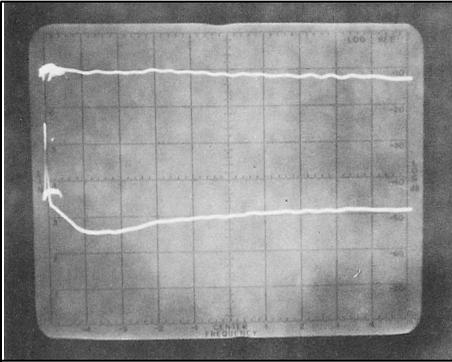


Fig 8: Photograph of the author's prototype from the other side.



**Fig 9: Directivity of the directional coupler bridge constructed as Fig 6 with two precision terminations manufactured by HP. Measured between 0 and 1GHz (H: 100MHz/T; V:10dB/T)
2MHz: -22dB, 4MHz -30dB,
10MHz -36dB, 1400MHz -36dB**

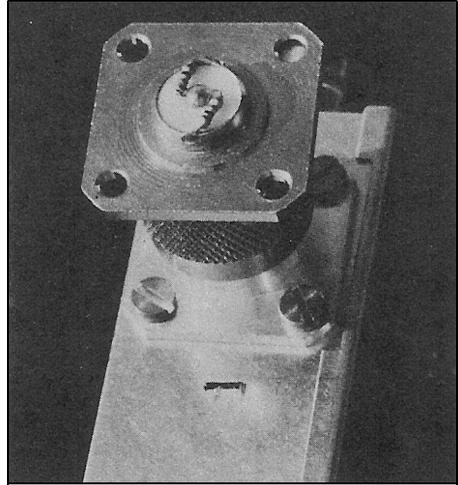


Fig 10: Photograph of a homemade termination using an N flange connector and two 100Ω resistors.

36dB, 1400MHz: -36dB. The lowest cutoff frequency of the bridge can be reduced down to below 1MHz by using a longer case and thus a longer balun.

9.

Components

- 1 metal case
- 2 BNC flange connectors
- 2 N flange connectors
- 2 N flange connectors for terminating resistors
- 1 PC board DJ7VY 006
- 8 resistors 100Ω / 1%
- 28 ferrite beads
- Approximately 20cm 50Ω Teflon cable SM50 or semi rigid copper cable

10.

References

- [1] Noise in Receive Systems, R Lentz, DL3WR, VHF Communications 7, Edition 4/1975 pages 217 – 235
- [[2] A Home Made Automatic Noise Figure Measuring System, M Dohlus, VHF Communications 15, Edition 1/1983 pages 66 – 83
- [3] A New Type of Preamplifier for 145MHz and 435MHz Receivers, M. Martin, DJ7VY, VHF Communications 10, Edition 1/1978 pages 30 – 36
- [4] Impedanz Meßbrücke CQ-DL 7/1982 pages 328 – 331
- [5] Using Vector Impedance Meters, Hewlett Packard: AN 86, page 11



Gunthard Kraus, DG8GB

Internet Treasure Trove

Research Publisher

If you are looking for something you can find it again here because there countless technology articles available sorted by subject.

Address: <http://researchpub.org/>

As an example: "dual band (1:4) Wilkinson power divider on Silicon".

Address: <http://researchpub.org/journal/je/number/vol1-no1/vol1-no1-1.pdf>

Microwaves101.com

This is an important address if you are interested in a particular area or topic of RF technology and are search for examples. There are numerous good links to each topic.

As an example: "Unstable Amplifier examples"

Address:
<http://www.microwaves101.com/encyclopedia/unstable-amplifier-examples.cfm>

Qucs

This is a somewhat exotic acronym that hides a fine thing, namely the free "Quite Universal Circuit Simulator". It is an "open source project" to simulate of as many problems in RF technology and

that means the cooperation of many people (mainly from higher education and University areas).

Of course the user interface is not so impressively designed as the commercially available "professional programmes" and sometimes the output results are not as perfect as in the professional. BUT: it is great that something like this is available. The "technical papers" are listed by theme with explanations that are very interesting. Just look or download the 500 pages of the powerful workbook in pdf format.

Addresses:

a) home page:

<http://qucs.sourceforge.net/index.html>

(b) user manual:

<http://qucs.sourceforge.net/docs.html>

(c) technical papers:

<http://qucs.sourceforge.net/tech/>

Hittite

This company is a well-known address for high quality microwave components. It is surprising that no application notes are to be seen when trawling the homepage until you realise the concept: they are generally associated with a module and therefore can be found in its description!

Also study the newsletter new products are presented there so it is important. An IC containing a PLL for the range of 33 to 4100MHz with VCO should serve to stimulate your appetite.



Website: <http://www.hittite.com>

PLL with VCO:

<http://www.hittite.com/products/view.html/view/HMC835LP6GE>

Radar Tutorial

This is a great thing: a thick and free tutorial with 10 parts. It goes from radar basics, through the technology up to the directional and the propagation characteristics of electromagnetic waves. Very well done, very interesting to read and it is available in three languages (English, German, French).

Address: <http://www.radartutorial.eu/druck/index.html#this>

JENS = International Journal of Engineering & Sciences

This website fully exploits the best search engines: all technical and research publications are catalogued here and archived. So what interests you?

Address: <http://www.ijens.org/index.htm>

An interesting Wi-Fi project as an example:

Address: http://www.ijens.org/Vol_13_I_01/137001-8282-IJET-IJENS.pdf

Daycounter Inc.

It is nice when companies install a set of online calculators for their customers on their homepage. There are for example: "complete RF amplifier design and analysis calculator" and other things.

Address:

<http://www.daycounter.com/Calculators/Complete-RF-Amplifier-Design-Analysis-Calculator.phtml>

Helix.air.NET.au

Sounds exotic but it is a very nicely presented page about the construction of Helix antennas.

Address: <http://helix.air.net.au/>

Queen's University, Gigahertz Integrated Circuits Group

Universities do much good, and here we have something from Ontario Canada. This enterprising group is very active.

Address: <http://post.queensu.ca/~saavedra/research/Main.html>

As example of the Java applets offered "lumped element filter Calculator":

<http://post.queensu.ca/~Saavedra/research/applets/FDA.html>

Antenna models for NEC and EZNEC

The author is "Cebik, W4RNL" know immediately as an expert. There is a collection of 800 models plus many articles and books. There are all the models for the NEC simulation that you need.

Of course the CD or the download cost money after fight your way through the huge list of offers and find what interest you.

Address: <http://www.antennex.com/Sshack/collect.htm>



VHF COMMUNICATIONS

A Publication for the Radio Amateur Worldwide

Especially Covering VHF, UHF and Microwaves

Volume No.45

Autumn

Edition 2013-Q3

Publishers

KM PUBLICATIONS,
503 Northdown Road, Margate,
Kent, CT9 3HD, United Kingdom

Tel: +44 (0) 1843 220080
Fax: +44 (0) 1843 220080

Email:
andy@vhfcomm.co.uk

Editor

Andy Barter G8ATD

**VHF
COMMUNICATIONS**

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

The 2013 subscription price is £22.80, or national equivalent. Individual copies are available at £5.70, or national equivalent each. Subscriptions should be addressed to the national representative shown in the next column. Orders for individual copies of the magazine, back issues, kits, binders, or any other enquiries should be addressed directly to the publishers.

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

© KM
PUBLICATIONS

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

Translated by Andy Barter, G8ATD with the assistance of Bing

Printed in the United Kingdom by: Printwize, 9 Stepfield, Witham, Essex, CM8 3BN, UK.

AUSTRALIA - Mark Spooner c/o, W.I.A. SA/NT Division, GPO Box 1234, Adelaide, SA 5001, Australia Tel/Fax 08 8261 1998
BELGIUM - UKW-BERICHTE, POB 80, D-91081 BAIERSDORF, Germany. Tel: 09133 7798-0. Fax: 09133 779833. Email: info@ukwberichte.com Web: www.ukwberichte.com

DENMARK - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

FRANCE - Christiane Michel F5SM, Les Pilettes, 89240 PARLY, France
Tel: (33) 03 86 44 06 91, email christiane.michel.s5sm@orange.fr

FINLAND - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

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

GREECE - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

HOLLAND - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

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

NEW ZEALAND - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

NORWAY - WAVELINE AB, Box 60224, S-216 09 MALMÖ, Sweden. Tel: +46 705 164266; GSM: 0705 16 42 66, email info@waveline.se

SOUTH AFRICA - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

SPAIN & PORTUGAL - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

SWEDEN - WAVELINE AB, Box 60224, S-216 09 MALMÖ, Sweden. Tel: +46 705 164266; GSM: 0705 16 42 66, email info@waveline.se

SWITZERLAND - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

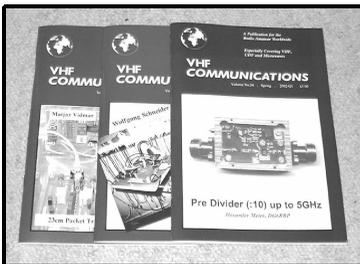
UNITED KINGDOM - KM PUBLICATIONS, 503 Northdown Road, Margate, Kent, CT9 3HD, UK. Tel: +44 1843 220080. Fax: +44 1843 220080. Email: andy@vhfcomm.co.uk

U.S.A. - ATVQ Magazine, Mike Collis, WA6SVT, P.O. Box 1594, Crestline, CA, 92325, USA,
Tel: (909) 338-6887, email: wa6svt@atvquarterly.com

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

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

ISSN 0177-7505



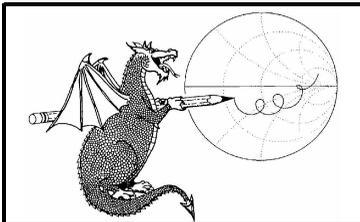
Back Issues

Available either as photocopies or actual magazines. Issues from 1/1969 to 4/2011 are £1.00 each + postage. Issues from 2012 are £5.40 each or £19.60 for all 4 issues + postage. See web site or page 34 of issue 1/2013 for back issue list.



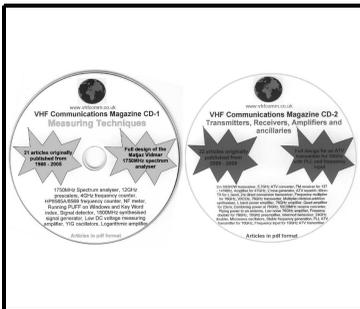
Blue Binders

A new style binder was introduced in 2010 with an embossed spine. 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 £2.20, Surface mail £3.25, Air mail to Europe £3.40, Air mail outside Europe £5.10)



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 £2.20, Surface mail £2.90, Air mail to Europe £3.20, Air mail outside Europe £4.50)



Compilation CDs

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

Each CD is £10.00 + postage. (UK £0.50, Surface mail £2.20, Air mail to Europe £2.70, Air mail outside Europe £3.30)



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, 503 Northdown Road, Margate, Kent, CT9 3HD, UK
Tel / Fax +44 (0) 1843 220080, Email: andy@vhfcomm.co.uk



HF, RF and microwave electronic components

Over 4000 items always ready in stock
shipped all over the world

www.rf-microwave.com

DIODES: Schottky, varicap, PIN, zero bias.
MIXER: balanced, active and passive.
TRANSISTOR: IF, RF, power.
GaAs-FET: low noise, power.
IC: wide band MMIC amplifiers,
logarithmic amplifiers, IF demodulators
for AM-SSB-FM receivers.
PLL and prescaler for synthesizer.
Regulators, RF power modules.
CONNECTORS: SMA, N, BNC and others
CABLES: teflon, semirigid, handyform.
CAPACITORS and **INDUCTORS.**
FERRITE CORES: wide availability of
toroids, binoculars, beads, rods, AMIDON.
FILTERS: IF, helical, saw, gigafil.
Terminations, attenuators, circulators,
switches, power splitters, VCOs,
metallic boxes, RF transformers.



Visit
the **SURPLUS** section
to find many
special offers
at affordable prices



ROGERS
CORPORATION

Multiplier and step-recovery diodes.
Power PIN diodes up to 1KW.
Noise diodes and noise sources.
Microwave prescaler dividers.
12Ω and 25Ω cables for matching of
power FET, 35Ω cables for power
splitters and Wilkinson combiners.
ATC100 high Q RF power capacitors.
Teflon laminates, ROGERS, RO4003.
Dielectric and puck resonators.
DC blocks, RF-microwave absorbers.
Fingers, Gigatrimmers, beam-lead.



Amidon



Special and hard to find
RF components



RF elettronica di Rota Franco - Via Dante, 5 - 20030 Senago (MI) - ITALY

Tel: +39 (0)2 99487515 Fax: +39 (0)2 99489276 E-mail: info@rfmicrowave.it Web: www.rf-microwave.com