

Band pass filter design

Part 9. Filter measurements

Richard Harris G3OTK

1. Introduction

So far we have considered the design of band pass filters. At some point we have to put theory into practice and build one. However, we will never have confidence in the design process if we are unable to measure the filters that we have designed and built. The equipment used to measure filters can range from a simple signal generator and oscilloscope, such as can be found in many shacks, through to a network analyser. The former will be of limited accuracy and dynamic range whilst the latter will usually be very expensive, although affordable units for the amateur constructor are becoming available. I have settled for an intermediate solution and have designed and constructed a synthesised signal generator and logarithmic detector.

For this article, we will design a three-resonator 7MHz band pass filter with 50 Ohm terminating resistors as a practical exercise. The filter will have a Chebychev response with 0.1dB ripple – this gives an almost flat pass band for most of the bandwidth – with a centre frequency of 7.1MHz and a 3dB bandwidth of 350kHz. There are two reasons for designing with 50 Ohm input and output terminating resistances. The first reason is to prove that we can match to a resistance of our choosing. The second reason is to maximise the dynamic range of the measurement – my equipment at the present time is limited to a 70dB dynamic range.

One important question is how complex can a filter be before it becomes necessary to use pre-set capacitors and devise an alignment procedure. This question is related to the tolerance of the components available. Another question is whether can off-the-shelf inductors with limited Qs be used or must toroids or air-core inductors be wound. The SPICE simulator (Ref. 2) that has been used for this series of articles can often be used to answer this question without having to build filters and measure their responses.

2. Measurement method

Fig. 1a and b shows the measurement set up. RF measurements are usually made using a signal source with a 50 Ohm output resistance and a detector with a 50 Ohm input resistance. The signal generator should have a low harmonic content – not easy if it is a simple LC oscillator. The output of the signal generator must be measured across the frequency range without the filter in place. The filter is then connected and the measurement repeated. From these two measurements, the insertion loss of the filter can be determined.

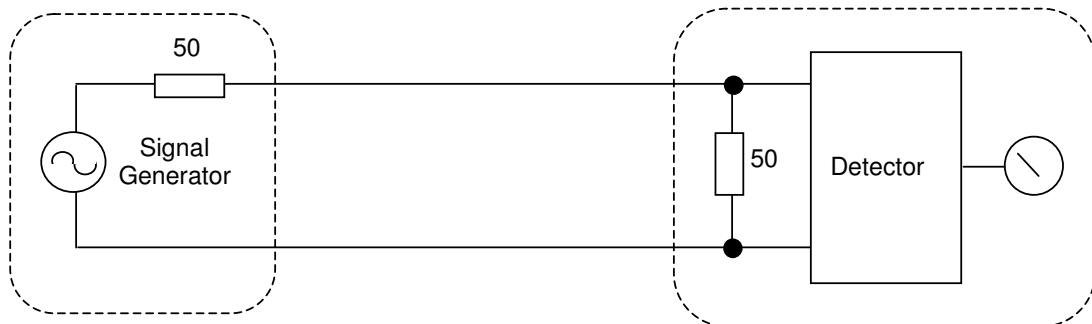


Fig 1a. Measuring the signal generator output

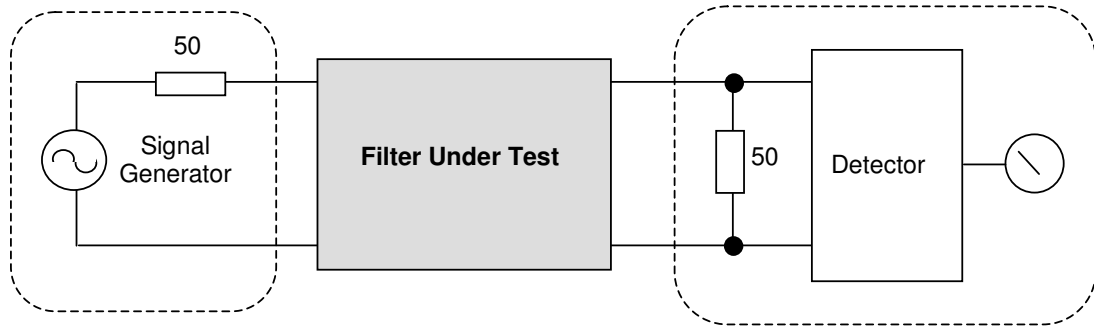


Fig. 1b. Filter insertion loss measurement

A high performance signal generator can be built around a Direct Digital Synthesis IC, such as the Analog Devices AD9834. The major part of the design work becomes software, not hardware. Useful facilities, such as linear and logarithmic sweep, constant output voltage and very low harmonic content, can be incorporated.

The detector can be an RF voltmeter, an oscilloscope or a logarithmic detector. This last detector is particularly attractive because the output can be scaled to give a direct reading of dB. It also has a large dynamic range. These days, such detectors are readily available as a single IC, for example the Analog Devices AD8307.

I built a signal generator and logarithmic detector around the Analog Devices ICs mentioned above. There are three modules (Fig. 2), a controller based around a PICAXE processor, a DDS module and a logarithmic detector. The controller sends information, such as the frequency and detector output via a serial connection to a PC, where it is read directly into EXCEL, using a data capture program, such as PLX-DAQ (Ref. 2).

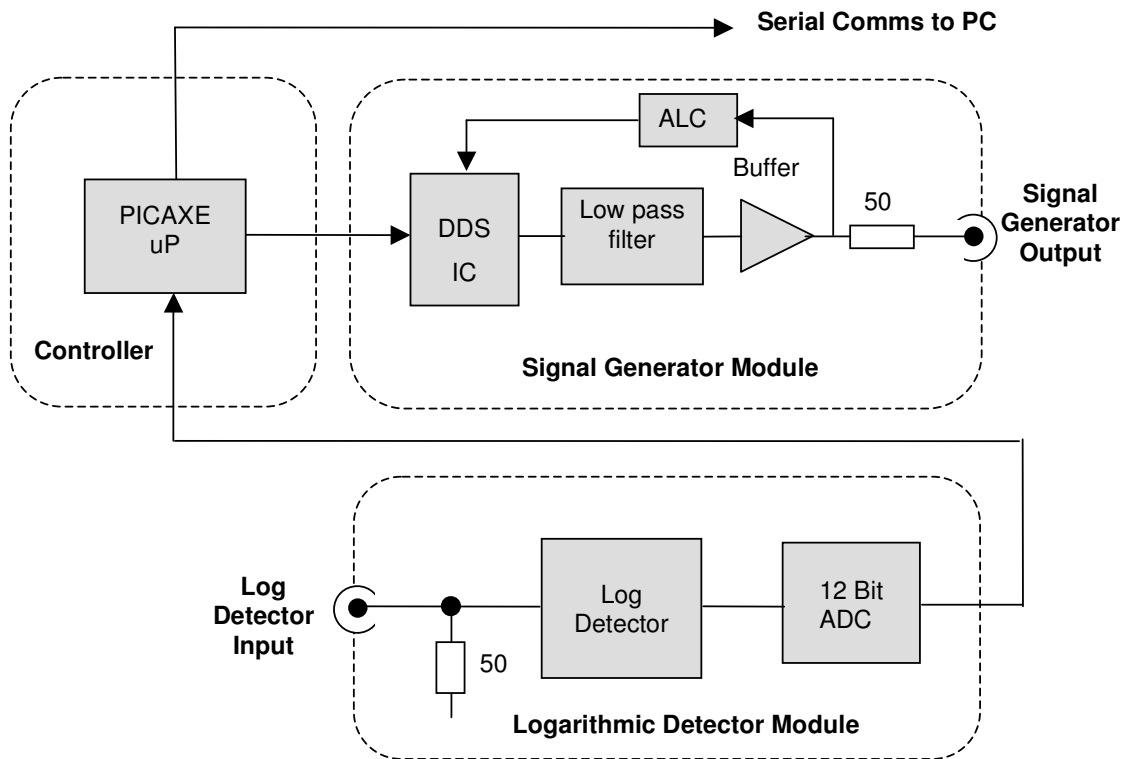


Fig. 2. The G30TK signal generator and logarithmic detector

Once in EXCEL, calculations can be made and graphs drawn and the output file of the SPICE simulator added to compare the measured with the predicted performance. A photograph of the controller and DDS module is shown in Fig. 3. The PICAXE processor (Ref. 3) is a PIC with an integral "PICAXE basic" interpreter and was originally intended for educational use. Interpreted languages are inevitably much slower than compiled languages, but speed is seldom an issue with a signal generator. The high level language is easy to use. No special programming hardware is required and the programming editor is a free download.

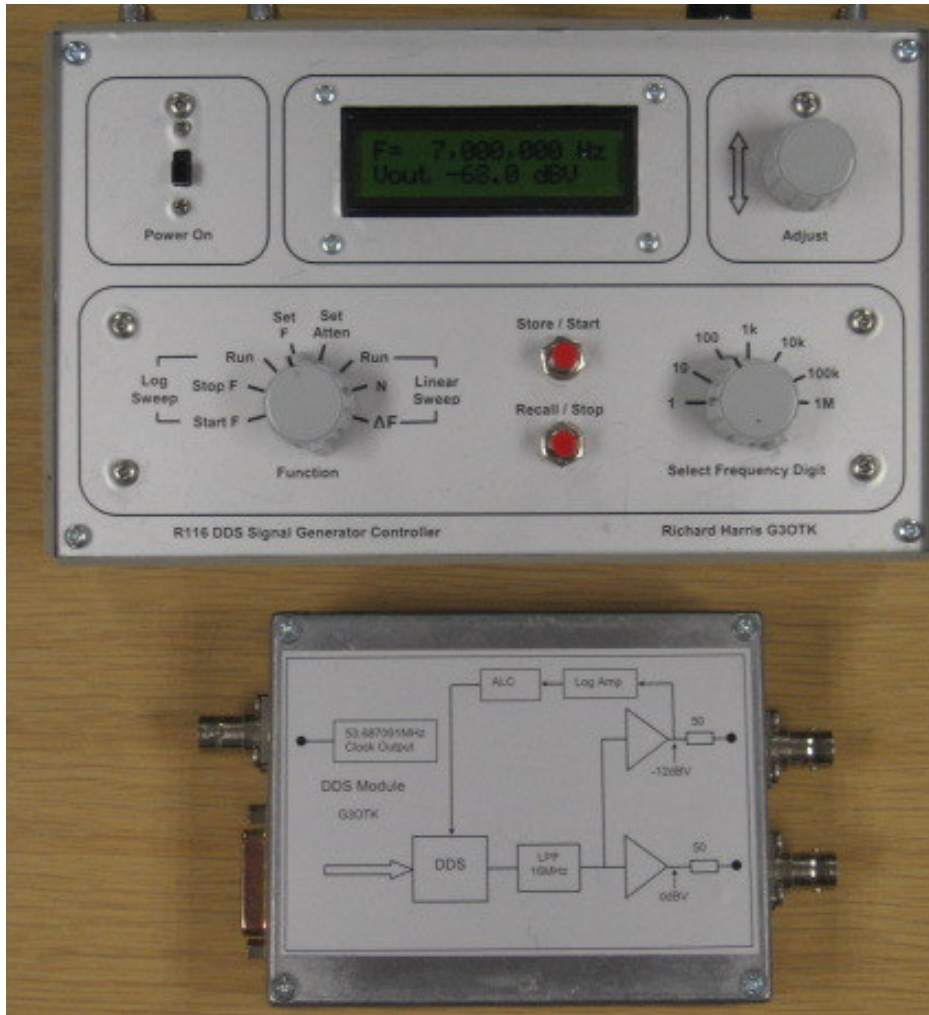


Fig. 3. Controller and DDS module

The DDS module and controller have the following specification:

- Frequency Range 10kHz to 15MHz
- Frequency steps 1 1Hz
- Harmonics <-45dB below carrier
- Output voltage 250mV (-12dBV) with ALC
- Clock frequency 53.687091 MHz
- Linear Sweep 1Hz to 50kHz/step
- Logarithmic sweep 100 steps/decade
- Memories 7

3. Experimental 7.1 MHz Filter

The specification for the filter is as follows:-

- Centre Frequency 7.1MHz
- Bandwidth (3dB) 350kHz
- Response type Chebychev 0.1dB ripple pre-distorted
- Inductors 2.27uH (23 turns of 22AWG on T50-6 toroid)
- Assumed inductor Q 200
- Input terminating resistance 50 Ohms
- Output terminating resistance 50 Ohms
- Capacitors NPO disc ceramic 5% tolerance

The inductance value is very much a “think of a number” exercise and may be determined by how many turns can be fitted onto the toroid with the enamelled copper wire to hand. From the Micrometals data for a T50-6 toroid core, the inductance for 23 turns is 2.12uH. Three cores were wound and the measured inductance (using an AADE LC Meter (ref. 1) was about 2.27uH.

A spreadsheet was compiled which took this specification and calculated the theoretical component values, as was described in Part 8 of this series. The circuit diagram is shown below in Fig. 4, drawn, for convenience, using the SPICE simulator.

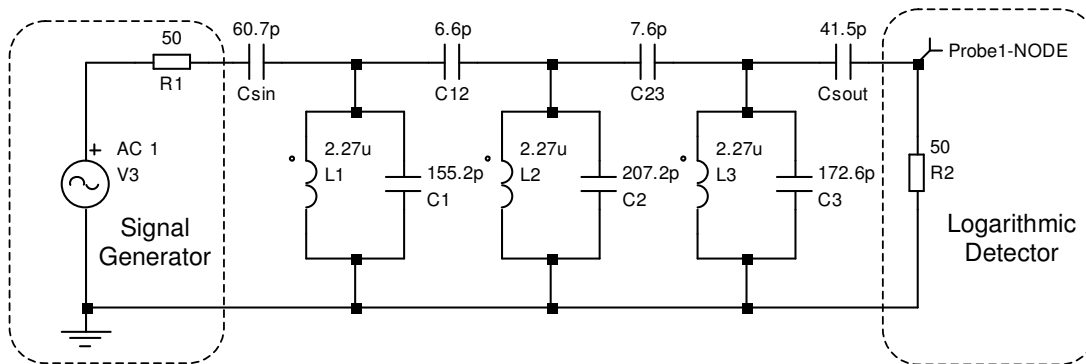


Fig. 4. Filter circuit diagram.

The question arose about whether the filter would need adjustment to give a satisfactory response or whether making up the required capacitances by connecting two or more capacitors in parallel would give adequate performance. Adjustment of complex filters is not straightforward and may be the subject of a later article. So the “paralleled capacitor” approach was adopted for the prototype filter. The values used are shown in Table 1

Circuit Ref	Value (pF)	Capacitors used (pF)
Csin	60.7	47 + 10 + 3.3 = 60.3
C1	155.2	150 + 5 = 155
C12	6.6	6.8
C2	207.2	150 + 47 + 10 = 207
C23	7.6	6.8 + 1 = 7.8
C3	172.6	150 + 22 = 172
Csout	41.5	22 + 15 + 4.7 = 41.7

Table 1. Capacitors used

With the exception of the 1pF, 4.7p and 6.8pF capacitors, all the capacitors were Multicomp (available from Farnell Electronics), 5% tolerance, +/- 60ppm/C temperature coefficient, low K, 50V, disc ceramics costing pennies. The 1pF, 4.7pF and 6.8pF disc ceramic capacitors were purchased from Rapid Electronics. No care was taken in the construction of the prototype filter, as can be seen in Fig. 5.

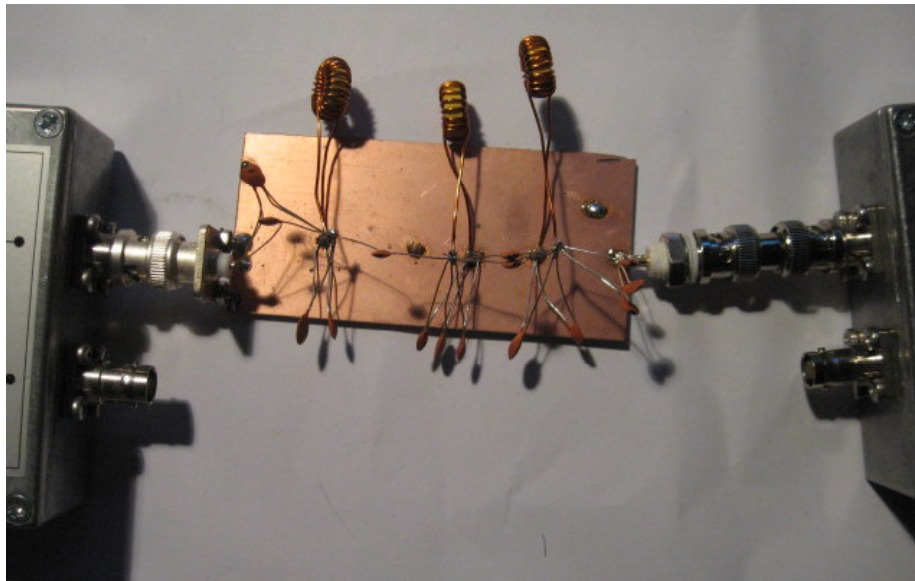


Fig. 5. Prototype filter with DDS module (left) and logarithmic detector module (right)

So, how did it perform? Rather well! The wideband response is shown in Fig. 6 and the pass band is shown in Fig. 7. It is compared against a simulation using SPICE (ref. 1)

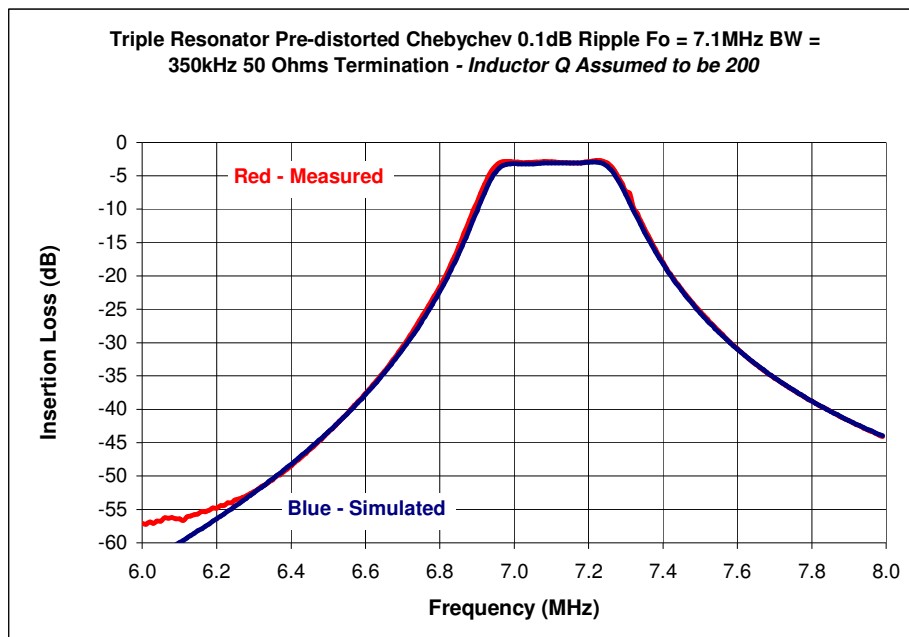


Fig. 6. Wide band insertion loss response

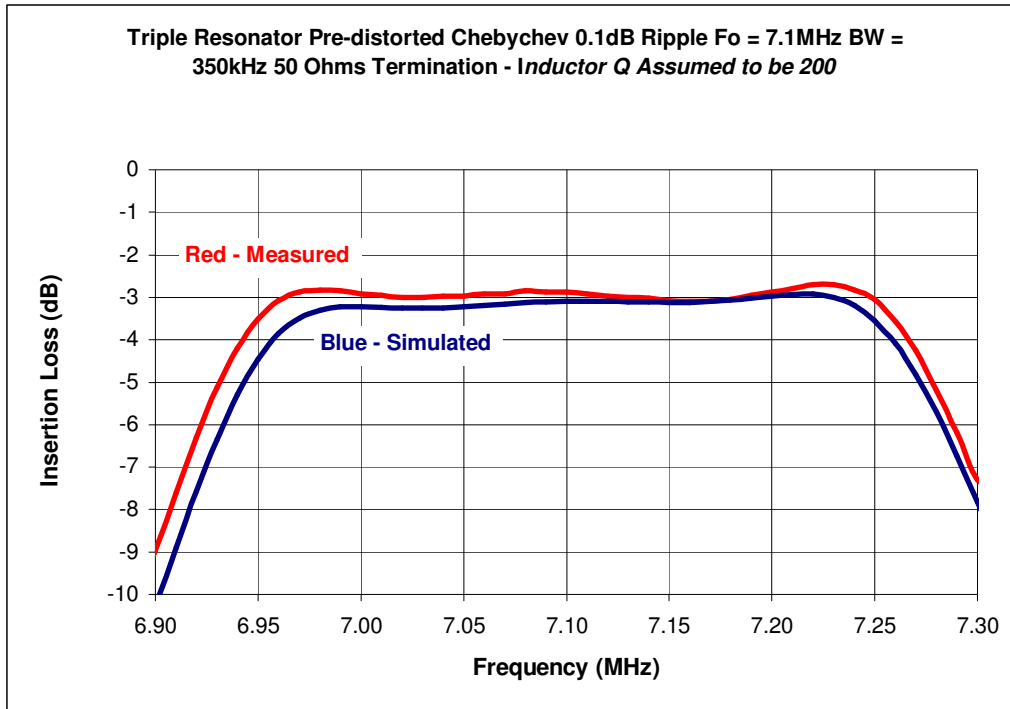


Fig. 7. Pass band insertion loss response

In Fig. 6, the measured response is starting to deviate from the simulated response below 6.2MHz. This is not due to a shortcoming of the filter but my test equipment running out of dynamic range. The pass band response in Fig. 7 is also a good match with the predicted response, being a fraction of a dB different from the simulated response. One reason is that the Q of the inductors may be higher than 200.

A useful piece of software was found on the internet (Ref. 2) that allows data to be sent directly to an EXCEL spreadsheet via one of the comm. ports of a PC. Prior to this, data was captured using HyperTerminal and then imported into EXCEL.

Firstly, the logarithmic detector module was connected directly to the DDS module and a sweep made of 200 steps at increments of 10kHz. This is the situation shown in Fig. 1a and gives a reference level at each frequency step. The filter was then connected (Fig. 1b) and a second sweep made. The difference between these two sweeps gives the insertion loss of the filter. The SPICE simulator was set up to give the same start and stop frequencies, with the same step size and the results copied and pasted into the EXCEL. The graphs were then drawn and annotated.

4. Conclusion

A three-resonator band pass filter for a centre frequency of 7.1MHz and 350kHz bandwidth was built and the insertion loss measured. The measurement was in close agreement with the simulated response using SPICE, giving confidence in the home made test equipment, the design process and SPICE as a means of accurately simulating filters.

5. References

1. www.aade.com. The AADE LCD meter has an accuracy of about 1% and can measure sub pico-Farad capacitors and sub-micro-Henry inductors. The AADE site also has an excellent filter design program as a free download.
2. www.simetrix.com – an excellent SPICE simulator
3. www.parallax.com/tabid/393/Default.aspx - this is free software than allows an external instrument to send data directly to EXCEL via a communication port of a PC. It doesn't run perfectly on my Dell laptop (running Windows 2000) for some reason, but is still usable.
4. www.picaxe.co.uk. The editor and manuals are free-downloads and the only investment required is the PICAXE processors (£2 to £5). Prototyping PCBs are available as well as other components, such as LCD displays.

Richard Harris G3OTK
June 2010
rjharris@iee.org

© R. J. Harris 2010