By Steven C. Hageman

Build Your Own Network Analyzer—*Part 2*

Put the Personal Network Analyzer to work and savor the circuit insight gained by actually *measuring* your designs!

In Part 1,[8] I discussed the design, development and construction of the Personal Network Analyzer (PNA). This month, we'll finish the setup. I'll briefly discuss the software and show you some of the ways you can put this useful tool to work.

The PC Control Program

The PC control program, *ANALYZER.EXE*[9] gives the hardware its instructions as to how to perform. The basic sequence is as follows:

• Setup for a sweep, get the start and stop frequencies and number of points to be swept.

• Get any other information about hardware external conditions that the hardware can't determine on its own (ie, is a ×10 probe connected to an input?). Also, set the minimum and maximum source-amplitude limits if needed.

• Start the sweep.

1. Send a frequency word to the source.
2. Autoscale the source and receiver attenuators to keep the dynamic range optimized (if needed).
3. Make the receiver readings; repeat if not settled.
4. Repeat for all frequencies in the sweep range.

• Plot the sweep data (ie, make a Bode plot).

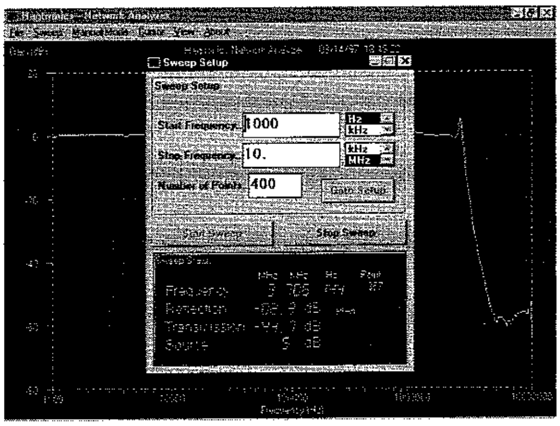
• Allow the user to view, measure (using cursors), zoom and print the resulting Bode plot.

ANALYZER.EXE has a continuously updating, single-frequency Manual Mode, which allows peaking, nulling or tuning the network under test in real time.

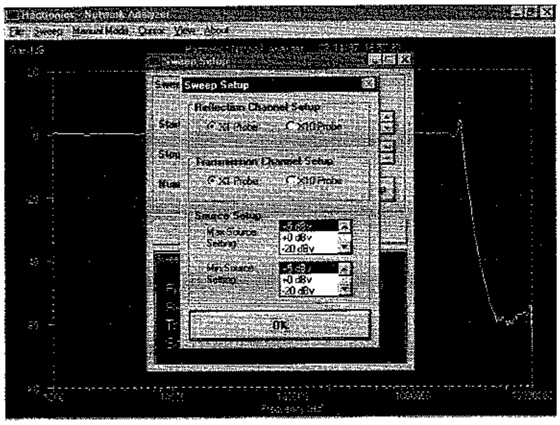
The program is written in 16-bit, Visual Basic for *Windows* and runs under *Windows* 3.1 and *Windows 95*. Visual Basic is perhaps the easiest way yet devised to program under *Windows* and allows easy graphics programming, which is a large part of this program.

The program is written in three basic modules, the first being the low-level hardware control. This is where the code communicates with the hardware via the serial port to do such things as set the DDS frequency,

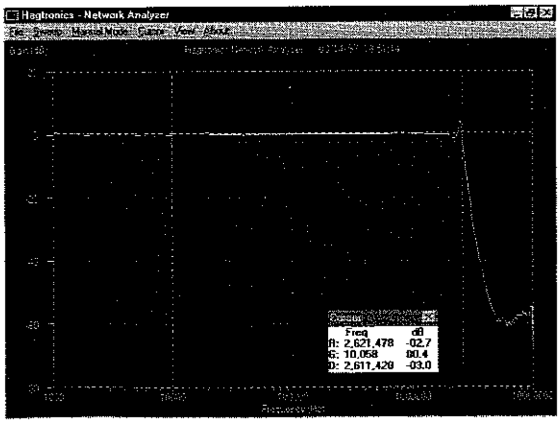
[8]Notes appear on page 39.



Before a plot can be made, the analyzer must know which frequency range to sweep and the number of points to sweep. The black window provides information on how the sweep is progressing. You can stop the sweep at any time by pressing the **Stop Sweep** button. Frequencies can be entered in engineering units by entering say, 10, then selecting the MHz spinner after the number for 10 MHz.



The **Sweep Setup** submenu allows greater adjustability of the sweep parameters. You can compensate for the use of ×10 probes, to ensure the measured decibel values are correct. The source's minimum and maximum values can be set during a sweep by using the spinners. When measuring a high-gain (or loss) circuit, these set the minimum and maximum source amplitude that the sweep will use. To maximize the receivers' dynamic range, the program automatically adjusts the source's amplitude (within the limits set) during a sweep.



Here is the result of all your hard work! This plot shows the response of a 5 pole, 2.5 MHz low-pass filter. Also shown are the cursors (the vertical dashed lines) that allow measurements to be made *on the plot*. The cursor window displays the red cursor's frequency and amplitude (**R**). There is also a green cursor (**G**). The delta frequency and amplitude between the red and green cursors is shown after the **D** (for delta). The cursors shown here are placed to measure the –3 dB points of the filter's response.

Table 1
Guidelines for Using the ×10 and ×1 Probes

Circuit Under Test	*Reflected Setting*	*Transmission Setting*
High-level passive or active circuit, with very little gain or loss, ie, < 20 dB	×10	×10
Low-level passive or active circuit, with very little gain or loss, ie, < 20 dB	×1	×1
High-impedance circuits ie, > 2.5 k Ω	×10	×10
Passive filter or network with net loss	×10	×1
Active network with a gain >20 dB	×1	×10

These guidelines will help you optimize the network under test for dynamic range. They will also help keep the receivers free of underload or overload.

make an A/D reading, or switch an attenuator in or out.

The next layer is the high-level hardware control where actual frequency sweep and autoscaling of the hardware attenuators takes place. The result of this layer is a data array of frequencies and decibel ratio values for later display.

The last layer is the graphics display. The functions here control how the data is displayed, allow zooming in on the data and manipulate the on-screen cursors.

Using a PC as a controller has the advantage of nearly unlimited number crunching ability. This is quite handy when we want to convert nonlinear or slightly nonlinear functions to linear ones for display. The nonlinear function I'm referring to here is the RSSI output of the NE/SA604 IC. Just because the chip has some nonlinearities doesn't mean that we have to live with them. Using a set of precision attenuators, I found the actual RSSI output for a known input over an 80 dB range of operation in 5 dB steps. I then wrote a function for the program that uses the measured data and linearly interpolates between points to improve the linearity to less than 0.4 dB.

As for speed, the program is pretty much limited by the RS-232-C transmission time. On any computer faster than a 33 MHz 80386, the RS-232-C communication time—not the computer speed—is the limiting factor. On my development system (a 50 MHz 80486), the program can do slightly better than 100 frequency points per minute.

Adjustments / Performance Checks

The only circuit adjustment required is that of setting the RSSI full-scale output. This is most conveniently done by using the PC control program in the Manual Mode and connecting the receiver inputs to the **SOURCE OUTPUT**; (use a 50 Ω termination at J3 of Figure 4). Adjust the source for an output of −20 dBV at 100 kHz. Set the receiver attenuators to **OFF**. Then set R3, **GAIN ADJ**, (through the receiver's shielded enclosure) to deliver exactly 230 A/D counts on both receiver channels.

That done, remove the covers from both receivers and measure the voltage at the bottom end of the 100 kΩ RSSI resistor, R2. This voltage should be around −1.5 V and be the same for both receivers. If the voltage is much more negative, it suggests that the noise floor is too high. A high noise floor is almost always caused by improper shielding around the NE604. Work on the shielding and make the gain adjustment again until the desired results are achieved.

Optimizing the Analyzer's Dynamic Range and Fabricating ×10 Probes

I modified a set of old ×10 scope probes for use with the input impedance of this circuit. First, I removed the 1 MΩ resistor in the probe tip and replaced it with a short piece of wire. I then gutted the compensation circuit in the small housing at the scope end of the probe. I drilled a hole for a SPDT toggle switch in the compensation circuit body and added a 90 kΩ metal-film resistor from the probe tip wire to the BNC scope output. The switch selects ×1 or ×10 attenuation by simply shorting out the resistor in

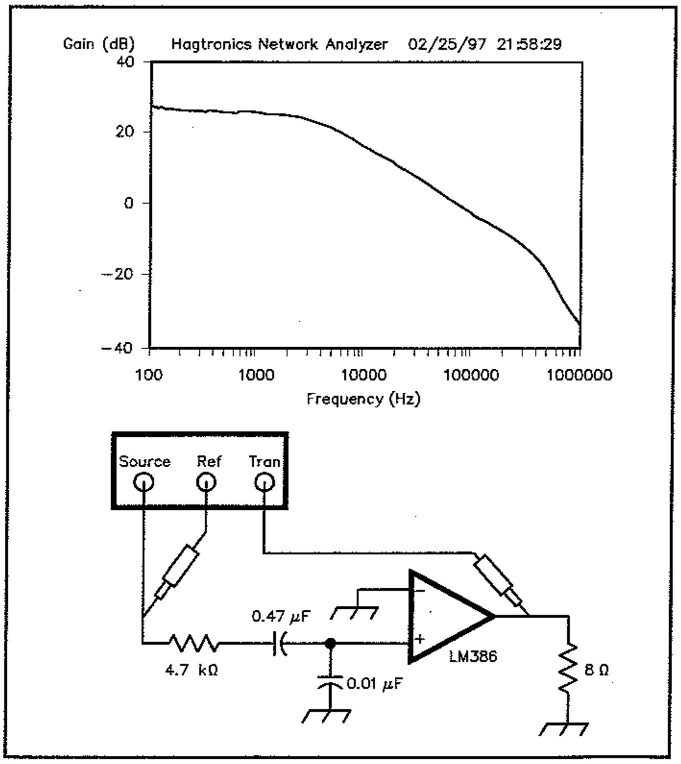


Figure 6—Measured response of an audio amplifier using the popular LM386. The response clearly shows the dc gain of 26 dB, the 3 kHz filter on the LM386's input and the LM386's pole frequency of 300 kHz.

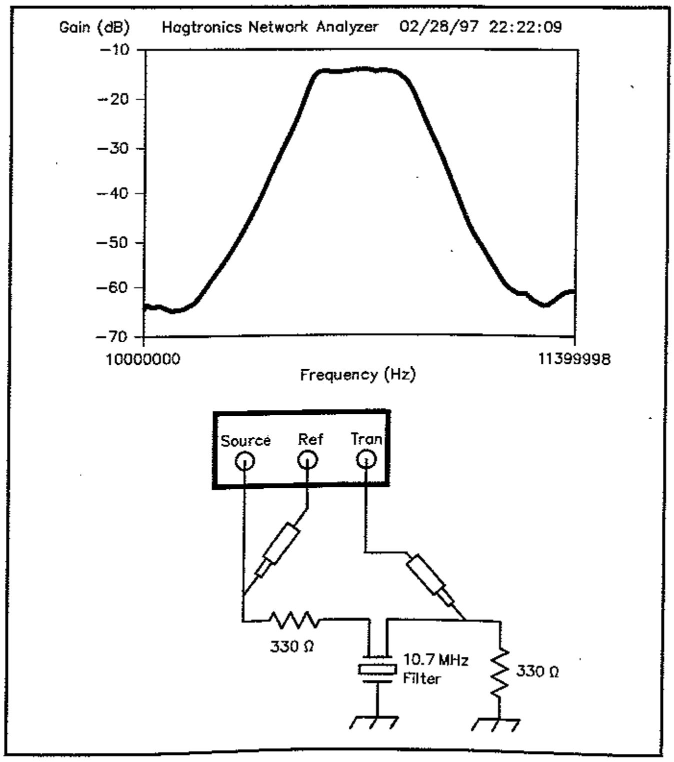


Figure 7—Ceramic IF filters are popular in 10.7 MHz IFs. Here, a 300 kHz-wide filter is measured with the PNA. The results were compared with the measurements of an HP4195 network analyzer. The HP4195 measured an insertion loss of 13.9 dB at 10.7 MHz and a 6 dB bandwidth of 360 kHz. The PNA reported a nearly identical insertion loss of 13.8 dB and a 6 dB bandwidth of 353 kHz.

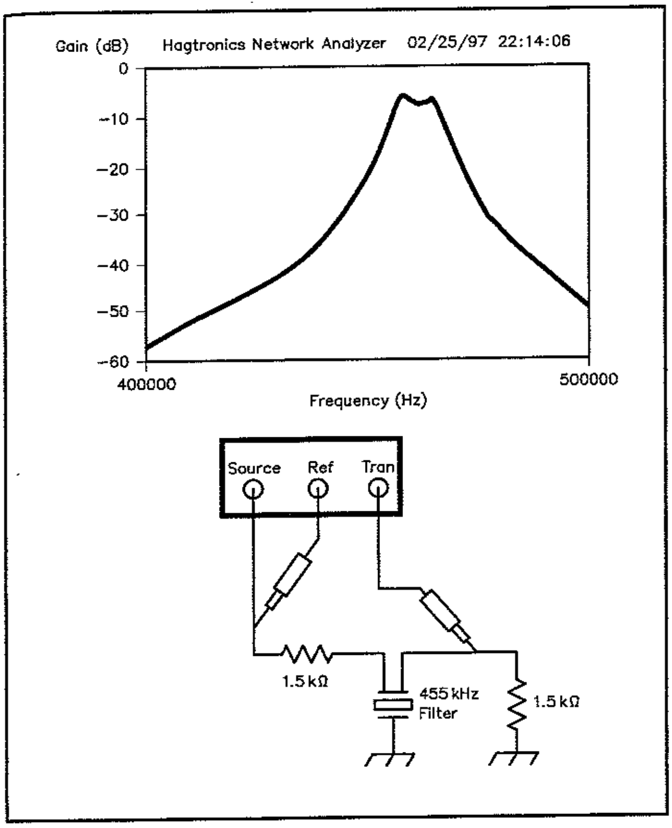


Figure 8—455 kHz ceramic filters are popular in radio circuits. Here, a 12 kHz-wide TOKO filter is analyzed. The double, tightly coupled (or overcoupled) hump response is plainly visible. When measuring filters, be sure to terminate the filter in the manufacturer's recommended impedance to ensure accurate measurement.

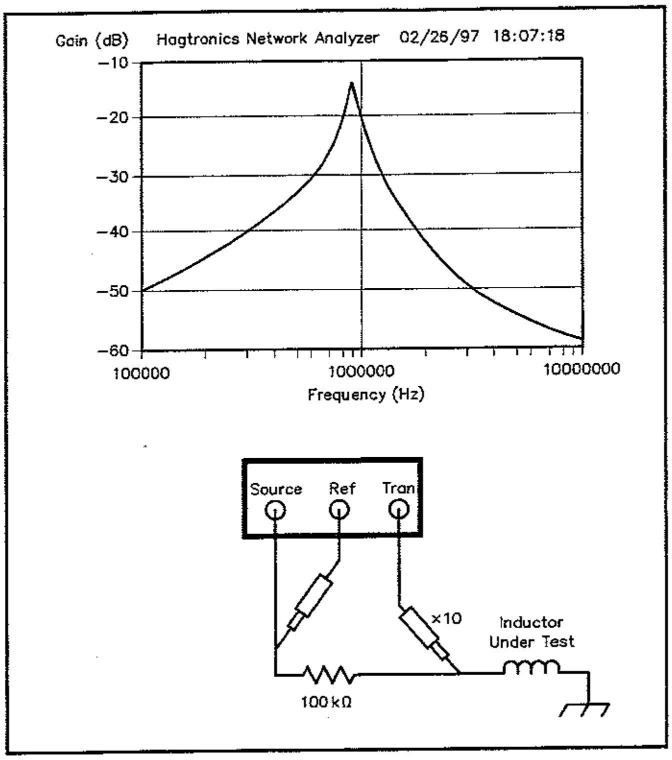


Figure 9—Inductors can be analyzed easily. Here, a 270 μH choke is checked. The source resistor was increased to 100 kΩ. This, combined with a ×10 probe, gives an equivalent input impedance of 50 kΩ. The self-resonant frequency is shown to be 900 kHz. The inductor value can be calculated as follows: The frequency at –40 dB is approximately 300 kHz, this corresponds to a reactance of 505 Ω. Using the formula $X_L/(2\pi f) = L$, gives a measured inductance of 268 μH at 300 kHz.

the ×1 position. In the ×10 position, the input impedance of the network analyzer is increased to 100 kΩ (90 kΩ in the probe added to the 10 kΩ input impedance of the analyzer) and the attenuation is increased to ×10 (20 dB).

Having good, shielded cables from the reworked scope probes to the network under test greatly helps to reduce the amount of noise that may otherwise be picked up by the receivers. When the ×10 switch position is used, large-amplitude signals can be measured easily (see Table 1).

The *ANALYZER.EXE* setup screen allows the probe gain to be taken into account, automatically offsetting the data by the probe(s) attenuation factor.

With very high gain amplifiers, the source may need even more attenuation than is possible with the built-in source attenuators alone. In these cases, another 20 dB (or more) may be needed directly at the **SOURCE** output to keep the amplifier operating in the linear region. Nonlinear amplifier performance can be identified by the unexpected appearance of an apparent "gain dip" in the circuit under test during a sweep. If nonlinear circuit performance is suspected, rerun the sweep while monitoring the circuit with an oscilloscope to be sure that the circuit is not saturating.

If the network under test is saturating, you may be able to cure it by adjusting the **Maximum Source Output** to a –20 or –40 dBV value in the *ANALYZER.EXE* program setup screen.

Some Measurement Examples

Here are some examples of measurements you can make with the PNA. Use your imagination and creative skills to devise other techniques. We'll start with something simple.

Audio Amplifier Analysis

Figure 6 shows how I tested the frequency response of an audio amplifier circuit using the venerable LM386 with an 8 Ω speaker, and setting the **SOURCE** output to –40 dBV. The amplifier has a 10 Hz to 3 kHz bandwidth that is set by the input RC filter. The low-frequency gain of 25 dB is just what the data sheet says it should be. The high-frequency roll-off at 300 kHz is the first pole frequency of the LM386 itself, adding to the input RC pole for a two-pole total roll off. Again, just as the data sheet predicts.

Ceramic IF Filter Analysis

Ceramic filters are used extensively in superheterodyne receivers. The PNA can analyze most common types of 10.7 MHz and 455 kHz filters (Figures 7 and 8).

I first checked a 10.7-MHz filter. These small, three-terminal ceramic filters have a 330 kHz (nominal) bandwidth. Measurement results obtained with the PNA were compared to those made with a Hewlett-Packard 4195A network analyzer. Using the HP4195A, the filter's insertion loss measured 13.9 dB; the PNA measured it at 13.8 dB. The filter's 6 dB bandwidth measured 360 kHz using the HP4195A; the PNA measured it as 353 kHz, an error of slightly less than 2%.

Next, I analyzed a 455 kHz IF filter. This was a 12 kHz ceramic filter and the double-tuned hump can be seen clearly.

Both filter-measurement setups used a –20 dB **SOURCE** level to prevent overloading the filters. Direct coaxial cable connections were made to the receivers. The input and output impedances of the filters were matched to the filter manufacturer's specifications (330 Ω and 1.5 kΩ, respectively) using 1% tolerance resistors.

Measuring Inductance

You can measure inductors over a 1 kHz to 16 MHz range. For this, the **TRANSMISSION** port probe is set to the ×10 value to increase its input impedance to 100 kΩ, and the source resistance is increased to 100 kΩ: The resulting R_{source} is 50 kΩ. The plot in Figure 9 shows a 270 μH choke's impedance versus frequency. The dc resistance (RDC) of this choke was 14 Ω which, at –73 dB, is well out of the dynamic range when using a 100 kΩ source resistor. The choke's self-resonant frequency at 900 kHz is plainly visible in the plot. The inductor value can be calculated by picking any convenient

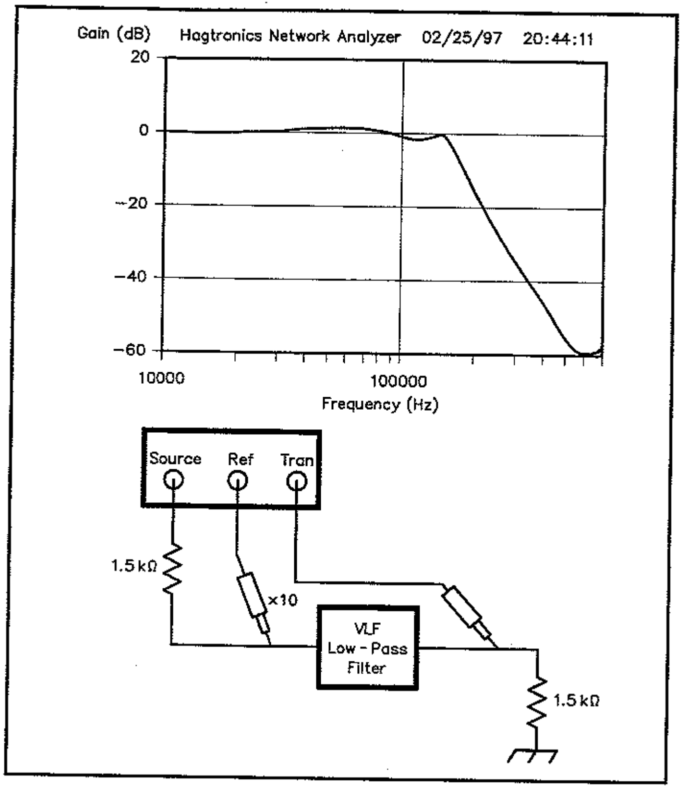


Figure 10—The Personal Network Analyzer measures a low-pass filter designed for a VLF receiver. The passband ripple is readily measured using the control program's built-in cursors. The analyzer dynamic range of this measurement is about 60 dB. The schematic shows the connections that are made between the analyzer and filter. The source was set to 0 dBV for this measurement.

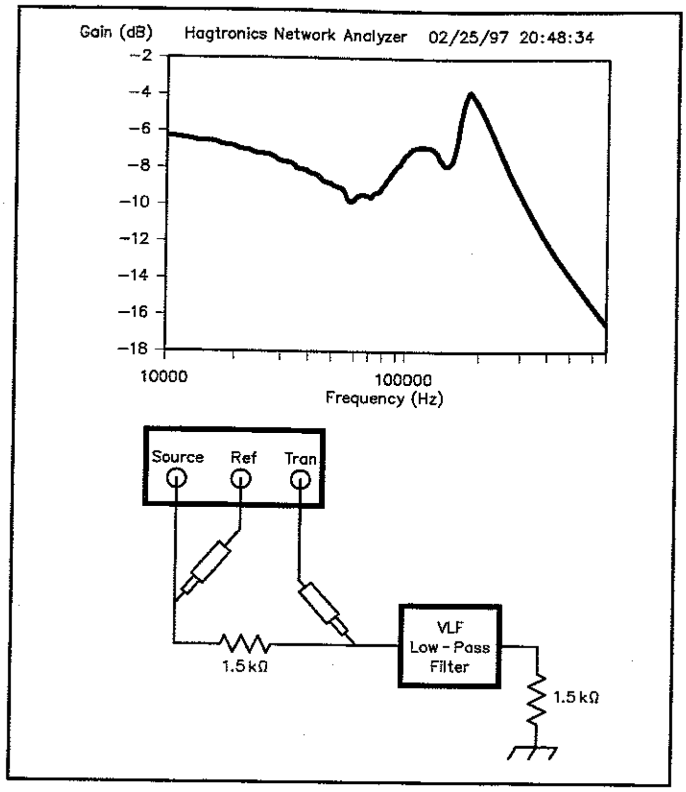


Figure 11—Network input impedance can also be measured. Here, the impedance of the VLF filter of Figure 10 is measured. The source resistance used is simply the terminating resistance of the filter. The low-frequency impedance is seen to be about 1.5 kΩ, the maximum impedance is 2.5 kΩ at 200 kHz, and at 700 kHz the impedance falls off to less than 300 Ω.

point on the upward slope. The –40 dB value is at about 300 kHz. The –40 dB response declares a reactance of 505 Ω. This amount of reactance at 300 kHz equates to an inductance of 268 µH.

VLF Filter Measurements

The passive Chebyshev filter in Figure 10 is designed for use in a VLF receiver front end. The filter impedance is 1.5 kΩ, so the receiver's 10 kΩ impedance won't appreciably load the filter. Because this circuit is a lossy network, the **REFLECTION** probe was set to ×10. The resulting Bode plot shows the network's response for a 400-point sweep. The plot clearly shows the 2 dB ripple in the passband and the stopband to –50 dB.

The input impedance of the VLF input filter can be analyzed by the circuit shown in Figure 11. This analysis simply uses the source-terminating impedance of the circuit itself. Several calculated impedance values are shown in Table 2.

For a discussion of return-loss bridges, which provide higher-accuracy impedance measurements, see *The 1997 ARRL Handbook*.[10]

Low-Frequency Tuned-Amplifier Analysis

Figure 12 shows how I checked the frequency response of a four-transistor, sharply tuned, 60 kHz band-pass amplifier. Because of the circuit's high gain, it needed extra

Table 2

Calculated Input Impedance of the VLF Input Filter of Figure 11

Frequency	*Input Impedance (Ohms)*
10 kHz	1500
60 kHz	700
190 kHz	2500
1 MHz	260

attenuation at the **SOURCE** output. I added a 60 dB pad between the PNA's **SOURCE** output and the amplifier input. A ×10 probe setting was used in the **TRANSMISSION** channel. I found the attenuator value by watching the sweep progress. When I realized that the amplifier was overloading, I reran the sweep with the pad installed. This amplifier also had an AGC circuit that had to be disabled to prevent interaction with the swept source. The plot shows the amplifier's nearly 80 dB gain and 150 Hz bandwidth.

ANALYZER.EXE was also used in Manual Mode with the source frequency set to exactly 60 kHz so I could peak each stage in real time.

Impedance Measurement

A network analyzer can also be used as a component or network impedance analyzer. If a known, fixed-value resistor is used in the source path as shown in Figure 13, the resulting network impedance can be calculated by the following equation:

$$R_{unknown} = \frac{R_{source}}{(1/\mathrm{Gain} - 1)} \qquad \text{(Eq 1)}$$

As an example, if the source resistor value is 100 Ω and the network under test is 100 Ω at some source frequency, the resulting gain will be –6 dB at that point. If at some other frequency the network has an impedance of 10 Ω, the resulting decibel ratio plotted is $-20.8\ \mathrm{dB} = (20 \times \log_{10}(10 / 110))$.

Capacitance Measurement

Figure 14 shows the resultant response of two types of 2.2 µF capacitors over a 1 kHz to 16 MHz range using a source resistor of 100 Ω. You can clearly see where the capacitor impedance flattens off at the capacitor's equivalent series resistance (ESR). The calculated ESR for the aluminum capacitor is approximately 4.4 Ω. The tantalum capacitor has a much lower ESR of 0.9 Ω.

Summary

The usefulness and circuit insight gained by actually *measuring* one's designs is incredible! I, for one, learn *much more* by designing circuits and then measuring how they *actually work* than by doing a "super-analysis" job up front. This is how I get a feel for building techniques and real-circuit parasitics that affect performance.

Technology is changing rapidly. I figure

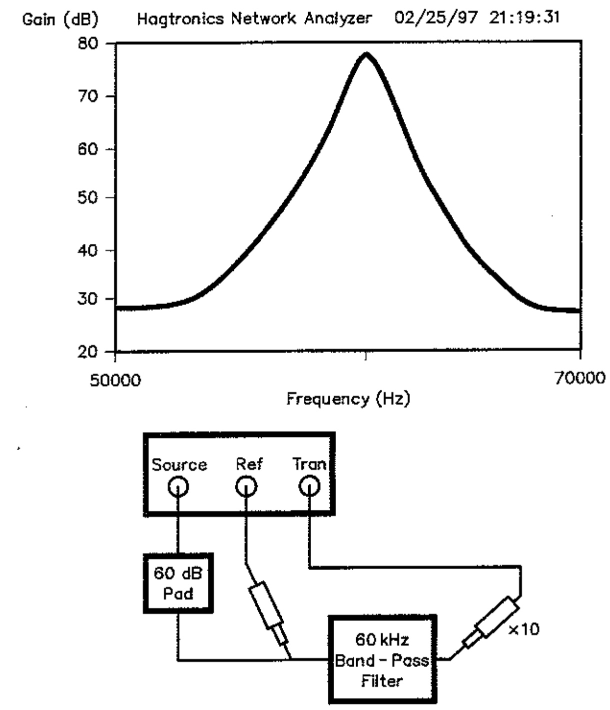


Figure 12—This is the response of a very high gain, multiple-stage, 60 kHz WWVB amplifier. This amplifier has so much gain that a 60 dB pad is used between the source and the amplifier input, to prevent the amplifier from saturating. The PC control program is used in a single-frequency Manual Mode to peak-tune the multiple LC filter sections in this amplifier.

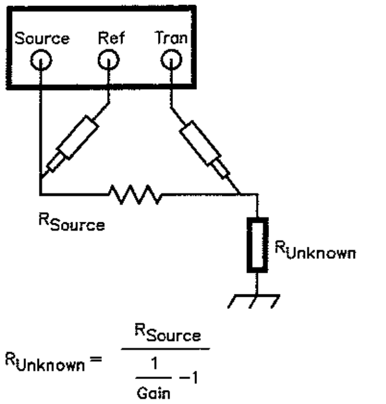


Figure 13—With the addition of a single resistor, a moderately accurate impedance measurement can be made using the PNA. Select a resistor value of approximately 10 times the maximum expected network impedance. Don't forget to take into account the network's 10 kΩ input impedance when measuring large impedance values. You can build a ×10 probe (see text) that increases the network analyzer input impedance to 100 kΩ, at the expense of only a little dynamic range.

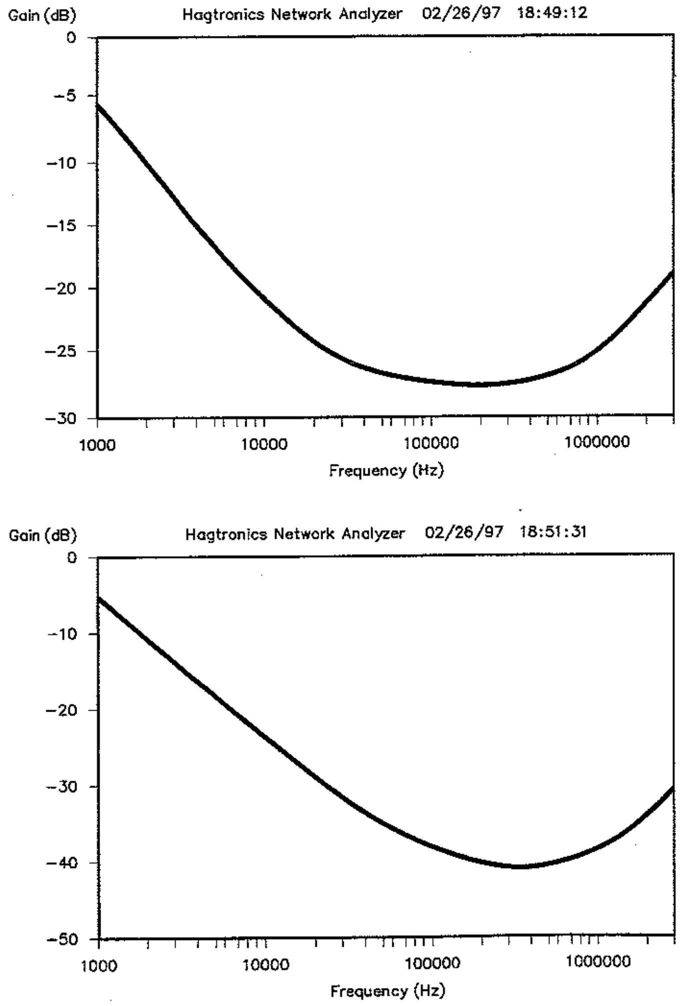


Figure 14—The results of measuring a pair of 2.2 µF capacitors of different types. The upper plot is that of an aluminum electrolytic capacitor; the lower plot is that of a tantalum capacitor. A 100 Ω source resistor is used. The equivalent series resistance (ESR) of the capacitors can be measured easily. If you ever wondered *why* a tantalum makes a better bypass capacitor than a standard electrolytic, the plots tell the story. The tantalum has a 5× lower ESR.

that in less than five years, my *next* network analyzer will be much more precise. (Be sure you stayed tuned to *QST* for that!—*Ed.*) I envision that single-chip DDS sources will be available with frequencies approaching the low VHF range (100 MHz?). By using two of these sources, a true tuned network analyzer could be built. This would entail using one DDS for the network source and using the second DDS source (programmed with a frequency offset) to drive receiver mixers. The receivers can then be made narrowband, which will greatly expand the dynamic range by lowering the noise floor. Linearity will also improve by having the RSSI circuit operate at a single frequency.

Having a narrow-band IF also opens the possibility of adding a phase detector to the IF limiter output stage. This would allow a true vector network analyzer to be built, and both gain and phase information could be displayed.

As the wireless revolution continues to drive IC performance, I envision logarithmic IF-strip ICs will become available. They'll have much more precise and linear RSSI outputs, perhaps better than 0.1 dB linearity. The level of integration will continue to increase, allowing the next generation of network analyzers to be built with the same number—or even *fewer*—components.

Notes

[8]Steven C. Hageman, "Build Your Own Network Analyzer—*Part 1*" *QST*, Jan 1998, pp 39-45.

[9]The software, including the source code, is in *HAGEMAN.ZIP* and can be found on the Internet (ftp to **oak.oakland.edu, dir pub/hamradio/arrl**) and on the ARRL BBS 860-594-0306.

[10]Paul Danzer, N1II, Ed., "Return Loss Bridges," *The ARRL Handbook for Radio Amateurs*" (Newington: ARRL, 74th edition, 1996), pp 26.41 to 26.42.