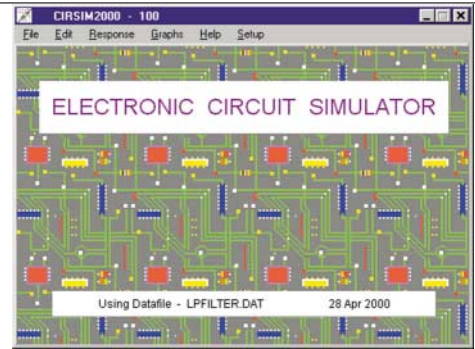## Special Review

CIRSIM SHAREWARE

COLIN SLOAN

A Circuit Simulation Program for the PC.

COMPUTER-BASED circuit simulation programs are now used extensively as an aid to circuit design. These programs do not actually design circuits but save time by enabling engineers to try out ideas by simulation before actually constructing hardware. There are some quite sophisticated packages on the market, such as Analyser III, MicroCap-V, Electronics Workbench and Tina Pro. The latter was reviewed in *EPE* last year.

These more expensive programs, which range in price from £100 upwards, are used in industry and by colleges for teaching purposes. They can handle a mixture of both analogue and digital circuits and even allow one to connect virtual instruments, such as an oscilloscope with appropriate controls, to view waveforms. A much cheaper but quite powerful simulation program called CIRSIM, which can analyse any linear analogue circuit, is now available and is the subject of this review.

ON ADMITTANCE

How do these programs work? Well, linear circuit analysis programs, such as CIRSIM, function by setting up a matrix of admittance values for each frequency at which the circuit response is required, these being derived from the component values defined for the circuit. It is essential to work with admittances rather than impedances (admittance is the reciprocal of impedance), as if there is no connection between two nodes of a circuit then the admittance of the path is zero, whereas the impedance would be infinity – a difficult value for computers to handle!

This matrix is known as the "indefinite admittance matrix" and relates the total current at any node in the circuit to the voltages at the nodes. The matrix is easy to set up as it exhibits a great deal of symmetry and would be of size $n \times n$ for a circuit containing n nodes. For three and four terminal components (such as transistors, op.amps and transformers), their own individual admittance matrices are derived and then added in to the main matrix – this is justified by the principle of superposition.

Having been set up, the complete matrix is reduced to a 2×2 matrix representing the two-port blackbox equivalent network of the circuit, from which the usual parameters may easily be derived; i.e., gain, phase shift, input and output impedances. For more details of the theory see the articles given in refs. 1, 2 and 3.
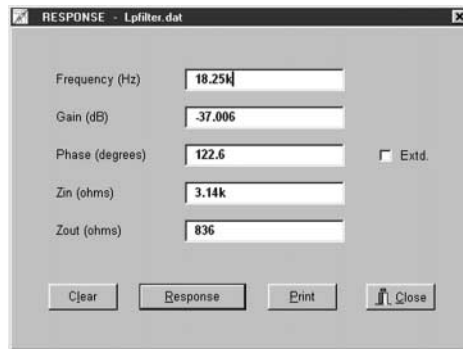

Fig.1. Results of analysis at a single frequency.

CIRSIM

CIRSIM itself is written in Pascal, using Borland Delphi, employing 32-bit code which will run under Windows 95, 98 and NT. It is capable of analysing circuits with up to 100 nodes and containing any mixture of resistors, capacitors, inductors, mutual inductors, transistors (both bipolar and f.e.t.), op.amps and thermionic valves. The high frequency parameters of both transistors and op.amps can be defined and there is no restriction on the total number of components.

In use a number is allocated to each nodal connection point on the circuit diagram (including earth) and then the component details and nodal connection numbers are input to the program. A simple text editor, with the usual cut and paste facilities, is incorporated in the program for this purpose. One can also add comments and circuit diagrams by using standard text characters, a facility which will be enhanced with appropriate circuit symbols in proposed later editions.

IN RESPONSE

Having defined the circuit, the simulator can calculate the response at any frequency. To do this, choose the appropriate menu item and type in the required frequency value. The screen will then display the gain in dB, the phase shift in degrees between input and output nodes, the input impedance and the output impedance at the specified frequency, as shown in Fig.1. With the 'Extd' box ticked, the results are displayed with much greater precision in floating point format.

The circuit definition is stored on disk so the analysis can be repeated at other frequencies or you can return to the editor to make circuit changes, adding or deleting components, to see how these affect the response. You can also change the input or output nodes so the response can be calculated at any intermediate point in the circuit. A listing of the component values and the results of the analysis can be output to the printer at any time.

It is possible to determine the stability criteria of oscillators by breaking the circuit loop at an appropriate point and finding that frequency at which there is zero phase shift between input and output. Provided the gain is greater than unity at this frequency, the circuit will oscillate when the loop is closed.

The simulator can plot the amplitude response of your circuit over a defined frequency range. Some 50 preset frequency scales are provided, ranging from 0·03Hz to 3GHz with each spanning 1, 2, 3 or 5 decades. A user-defined frequency scale is also available covering any range you care to define.

Progress is indicated by a horizontal bargraph while the results are being computed. On completion, another menu allows you to set the amplitude scale appropriately, after which the response is displayed on the screen in a 640 × 480 format.

After examining the result, you can return to the amplitude setting menu, revise the scale if required, and then display again. If you want a hard copy of the plot, a printer setup menu is provided which makes it possible to position the graph on the paper and to scale the plot. A typical result, Fig.2, shows the response of a low pass Chebyschev filter, and Fig.3 shows the response of a mutually coupled tuned circuit (as might be used in an i.f. amplifier) with various degrees of coupling; namely, undercoupled, critically coupled and overcoupled.

One unique feature of this program is that plots can be superimposed – a facility often not found in more expensive packages. Thus, one can plot and save a

response, return to the editor to make circuit changes, and then superimpose the revised response on the first – repeating this as many times as required.

The impressive result in Fig.4 shows the response of a seven-band graphic equaliser circuit. The various boost/cut potentiometers (sliders) were set to flat and the response plotted. The response was then plotted with each in turn set first to maximum and then to minimum positions. The resultant 15 plots were superimposed on one graph as shown.

It can also calculate the squarewave response of a circuit at any given frequency. This can be rather time consuming so to speed things up, with a slight loss of accuracy, one can restrict the number of harmonics used in the calculation. The squarewave response of the low pass Chebyschev filter plotted on the printer in the same way as the amplitude response is shown in Fig.5.

IN USE

CIRSIM is easy to install and use. It is robust, it doesn't crash if you type in silly values, and it appears to be accurate. The program designer claims to have compared the results obtained from CIRSIM with those using PCSPICE analysing the same circuit and found there was exact agreement.

It is very fast in operation with smaller circuits but slows up rapidly as the number of nodes increases. To analyse a circuit containing 100 nodes (which is quite large) takes about one second using a, now fairly ancient, 100MHz Pentium. As you would expect, graphs take longer to compute since 400 data points are involved.

Detailed documentation is provided which can be displayed on the screen using a simple menu system and more than 20 circuit examples are included in the package. These include simple active and passive filters, a loudspeaker crossover network, a cascode circuit, various audio amplifiers (including a valve amplifier), a Baxandall tone control, a gyrator and a 47-section transmission line.

The simulator is very useful for analysing linear circuits and can save a lot of time by resolving problems before actually building any hardware. At lower frequencies, up to about 10MHz, circuits perform as predicted by CIRSIM within the limits of experimental accuracy. At higher frequencies there are some discrepancies which are probably caused by inaccurate modelling of components – more especially stray capacitance and inductance not being included in the simulated circuit.

SUMMING UP

To sum up, in terms of accuracy and speed CIRSIM is as good as other more expensive programs designed to simulate linear analogue circuits while the graphical output is well presented with the advantage of superposition. It is also capable of analysing circuits containing thermionic valves, which is a bonus.

The program does have its limitations – it does not have all the bells and whistles of its more expensive counterparts, there are no facilities to import component data from files and it cannot handle logic circuits. However, for the price, just £10, it represents excellent value for money and is a good low-cost introduction to the art of linear circuit simulation.

A free evaluation version of the software, providing all the facilities of the program but handling a limited number of nodes, can be downloaded from the website **http://www.bells-hill.freeserve.co.uk**. This downloaded file contains full documentation and examples. The purchase price of £10 applies only to the version of the program handling the full number of nodes.

CIRSIM can be obtained from:

Peter Montgomery, Downings, Bells Hill, Stoke Poges, Slough SL2 4EG.

Tel: 01753 643384.

E-mail: cirsim@bells-hill.freeserve. co.uk ☐

References
1. Network Analysis and Synthesis by F. F. Kuo – J. Wiley and Sons.
2. Semiconductor Device Modeling for Computer-Aided Design by G. A. Herskowitz and R. B. Schilling – McGraw-Hill.
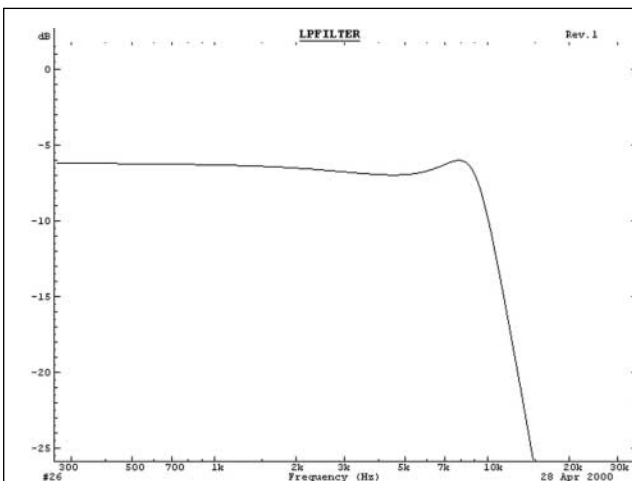3. Computer Circuit Simulation by W. Blume – Byte July 1986.

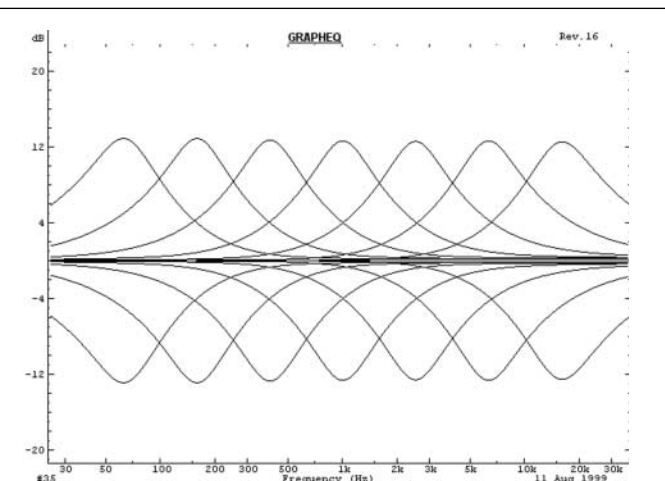Fig.2. Amplitude response of a low pass filter.

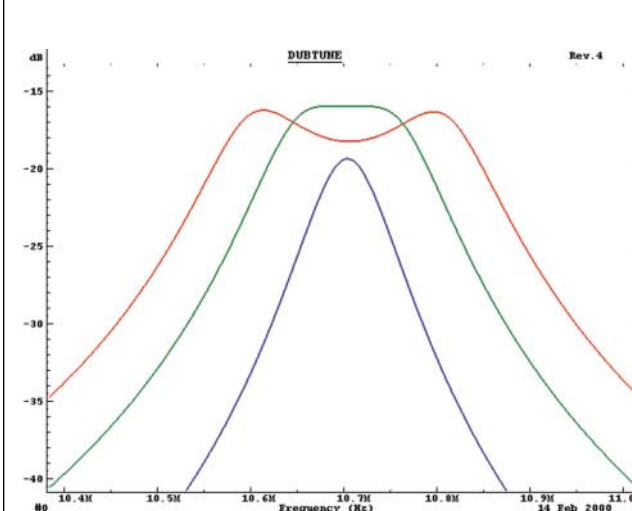Fig.4. Amplitude response of a seven-band graphic equaliser circuit.

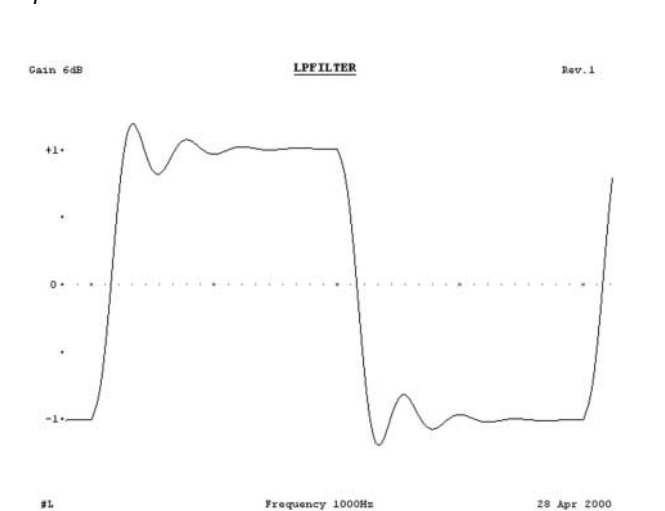Fig.3. Response of mutually coupled tuned circuits.

Fig.5. Squarewave response of the low pass filter, showing ringing around the cut-off frequency.