By Cyril Bateman

Simulating Inductors and Networks

Using the Micro-cap7 software, the author introduces a hands-on approach to SPICE circuit simulation to devise new, improved, user models, able to accurately mimic inductor behavior by frequency.

SPICE, the "Simulation Program with Integrated Circuit Emphasis," has become by far the most widely used circuit simulation program. It is provided with competent transistor macromodels, so performs well when simulating integrated circuits. Basic capacitor, resistor, and inductor models are also included, but only as idealized, theoretically perfect components, suited for use with the small values used when modeling integrated circuits, but far removed from almost all discrete, real-world components. This article shows how vastly improved, realistic inductor models can simply be produced using only the basic primitive elements found in every SPICE simulator.

Most SPICE analyses are made using large signal transient simulation to produce a time domain waveform of the circuit's behavior, just like probing the circuit using an oscilloscope. When modeling in the time domain, you can modify SPICE models to account for amplitude nonlinearity. You can also perform small signal AC, frequency domain simulations, and when using SPICE or a SPICE equivalent simulator, you can modify capacitor, resistor, and inductor models to also account for parameter changes with frequency. However, within SPICE, these frequency and amplitude dependent pa-

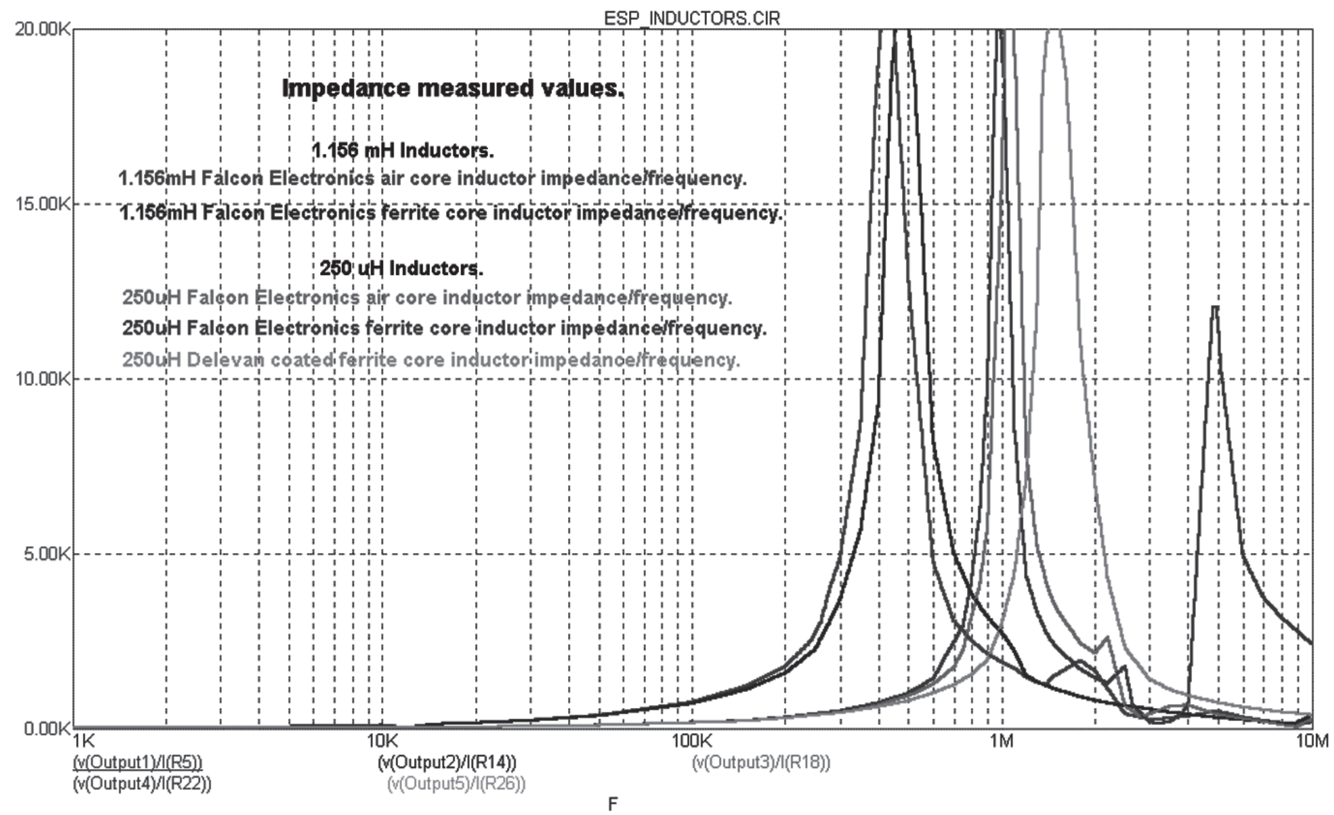

FIGURE 1: Measured impedance of low-loss inductors, both air cored and ferrite cored, designed for use in loudspeaker crossovers.

rameters are mutually exclusive; you cannot use both parameter sets within one simulation.

Of course, SPICE is not the only type of simulator; other simulators model principally in the frequency domain and many can combine both time and frequency domains together by using convolution as in the "Harmonic Balance" simulators. Such simulators are usually provided with a library of truly competent inductor models able to replicate real-world components, even to very high frequencies.

INDUCTOR MODELS

Unfortunately, few component makers provide realistic "SPICE" models, unless their components are specifically intended for use at high frequencies, for example, Coilcraft, which provides accurate simulation models for use in the "Microwave Office" software. As a result, simulations of larger value inductance using the basic "SPICE" model, even at modest frequency, result in large errors and misinterpretation of circuit behavior. Let me examine actual measurements of a few "real" inductors (**Fig. 1**).

All practical inductors include self-capacitance between adjacent wire turns and from the start and finish end-to-end turns. When using toroid cores, these end-to-end turn capacitances dominate unless you take care to leave sufficient unwound space, typically about 90° between start and finish winds. Next I examine typical "SPICE" plots using the default inductor models provided as standard.

The SPICE default inductor model (**Fig. 2**) agrees with my measured values at audible frequencies and continues to agree up to 100kHz. However, by 200kHz I find the 1.156mH inductor simulation suggests an impedance of 1454Ω, whereas the measured value was 1779Ω. Clearly these errors then increase rapidly with frequency. Many writers have used SPICE default models to simulate amplifier Nyquist response up to 1MHz, driving into a simulated loudspeaker cable and simulated crossover/speaker loads, using only basic SPICE models for cable and speaker, with invalid, unreliable conclusions.

Every inductor includes parasitic capacitance between adjacent coil turns and between start and finish windings. These capacitances result in the parallel resonances shown in **Fig. 1**. Inductors intended for use in loudspeaker crossover networks will exhibit a major resonance, typically between 100kHz and 2MHz; larger values resonate at even lower frequency. At higher frequency, as shown in **Fig. 1**, other lesser resonances should also be anticipated. For example, the 3.5mH inductors used in my horn-loaded speaker crossover resonated at 210kHz, but exhibited many smaller high frequency resonances.

AN IMPROVED INDUCTOR MODEL

Using SPICE basic components and with

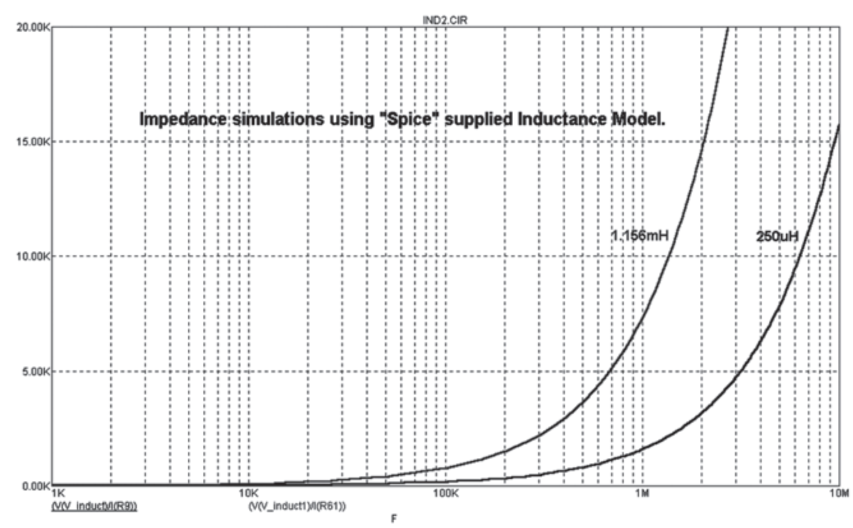

FIGURE 2: Simulated impedances of 1.156mH and 250μH inductance, same values as measured for *Fig. 1*, using SPICE default models.

a knowledge of an inductor's resonant frequency, you can easily devise a more appropriate computer model. You can calculate the effective self-capacitance using the equation:

$$C = \frac{1}{4\pi^2 F^2 L}$$

derived from the more common equation $F = \frac{1}{2\pi\sqrt{LC}}$.

Using the 400kHz resonance for the 1.156mH inductor (**Fig. 1**), you find its equivalent capacitance, the combined effect of its turn-to-turn and end-to-end capacitances to resonate at 427kHz must be 120pF. Using just these values results in too sharp a resonance, because the real inductor exhibits a measurable DC resistance of 2.184Ω; almost all is effectively in series with the inductance. Also because the resonating capacitances are complex, they will exhibit resistive losses; some loss will be in series with this capacitance, which affects the resonance width, together with a large shunt loss resistance value, which limits the peak impedance value.

These series and shunt resistances are best devised by running a few simulations, adjusting values as needed to approximate measured values. I find that a peak impedance shunt resistance of 40kΩ makes a good starting value. The measured DCR in series with the inductance, less about 0.1Ω for those lead wires which lie outside the main winding, provides an excellent fit to measured slope values, which can be finally tuned by adjusting a small resistance, in series with the shunt capacitance. These "cut and try" adjustments take little time, but how well do they work (**Figs. 3** and **4**)?

With a few extra components, you can add an additional "node" to also model secondary resonances.

Loudspeaker drivers can be treated as an inductive component, but for accuracy to high frequency they require several (many) additional nodes. With the impedance of a driver measured to about 10MHz, in addition to the usual audible frequency resonances which are usually well modeled, the drivers I measured ranged from an 18″ 250W Goodmans Power series with a peak impedance of 495Ω at 400kHz to a Kef B110 with peak impedance of 570Ω at 1.1MHz. In my ESP_replica network, the bass driver used measured 843Ω at 500kHz, and the T27 tweeter measured 825Ω at 2.4MHz. Within this frequency band, other drivers all measured as intermediate impedance peak values. With inductors typically peaking around 40kΩ, clearly the combined impedance of crossover and speaker driver at high frequency must approximate 5-600Ω when resonant, reducing with further increase in frequency.

I had two quite different crossover/speaker cabinets available. The first was my horn-loaded two-way system (**Figs. 5** and **6**), which measured a peak impedance of 525Ω at 900kHz. The other, the ESP replica assembly as used for my cable evaluations paper, measured a peak impedance of 575Ω at 1.5MHz.

At frequencies higher than this inductive resonant peak, the inductive leading phase no longer applies. The measured phase angles now lag, and the inductor impedance reduces with frequency, just like a capacitor. This often results in the circuit drawing unexpectedly high and capacitive currents.

MODEL DEVELOPMENT

The simplistic, idealized models for ca-

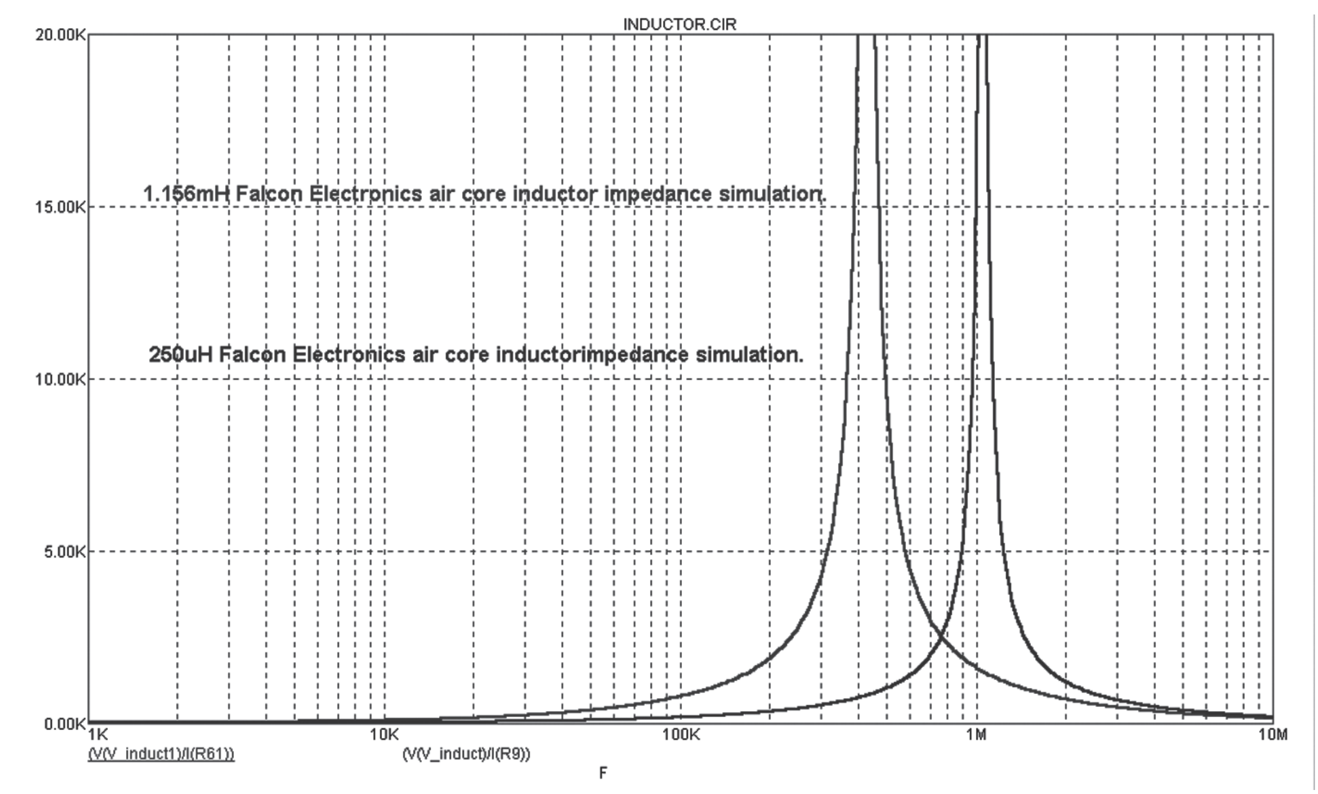

FIGURE 3: Simulations which closely approximate my measured impedance values for the first or main resonances of the 1.156mH, also, 250μH Falcon Electronics air core inductors, shown in *Fig. 1*.

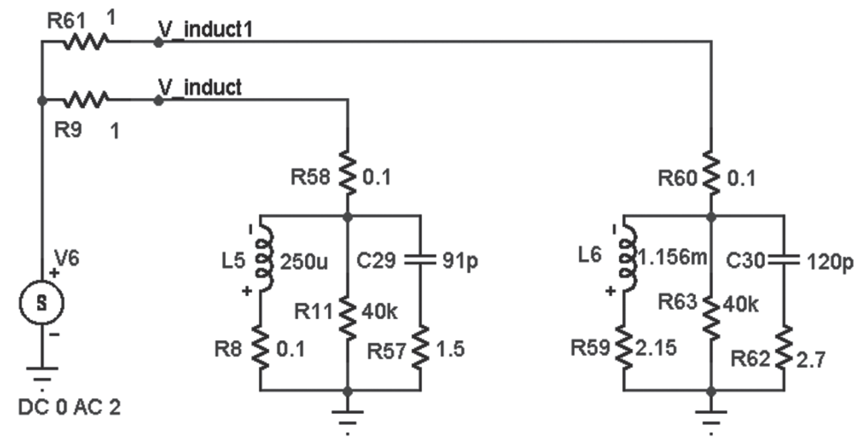

FIGURE 4: The simple SPICE models as used for the *Fig. 3* realistic simulations, of the first or main resonances for the 250µH and 1.156mH Falcon Electronics inductors.

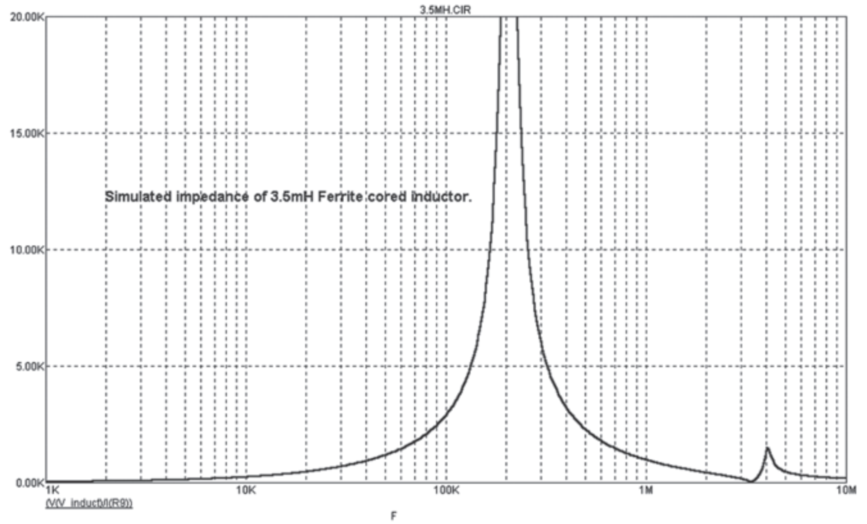

FIGURE 5: Simulated impedance of 3.5mH, ferrite cored inductor as used in my horn-loaded speaker crossover network. Actual measured peak impedance was 40kΩ at 210kHz.

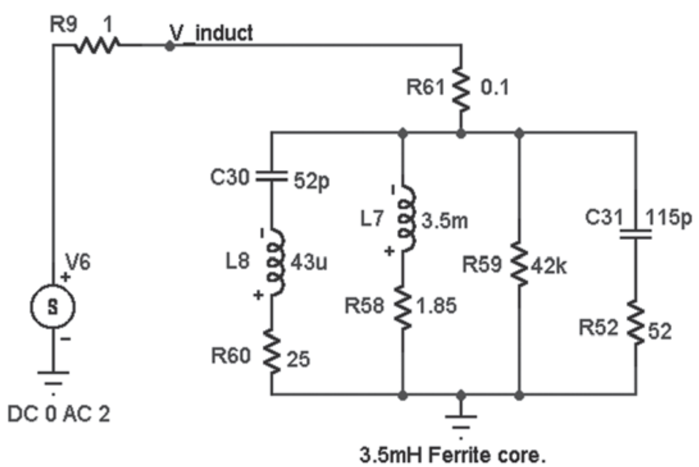

FIGURE 6: Model used for *Fig. 5* simulation.

pacitors, resistors, inductors, and transmission lines provided in most SPICE-based simulators can be useful for DC, transient, and low-frequency AC modeling; they do not, however, represent any practical components. In the real world, capacitors, resistors, and especially inductors, measure quite different from their SPICE predictions.

All components include parasitic elements, so are better described at least to moderate frequencies, by combining all three elements: capacitance, resistance, and inductance for each device. For realistic high-frequency simulation, every com-

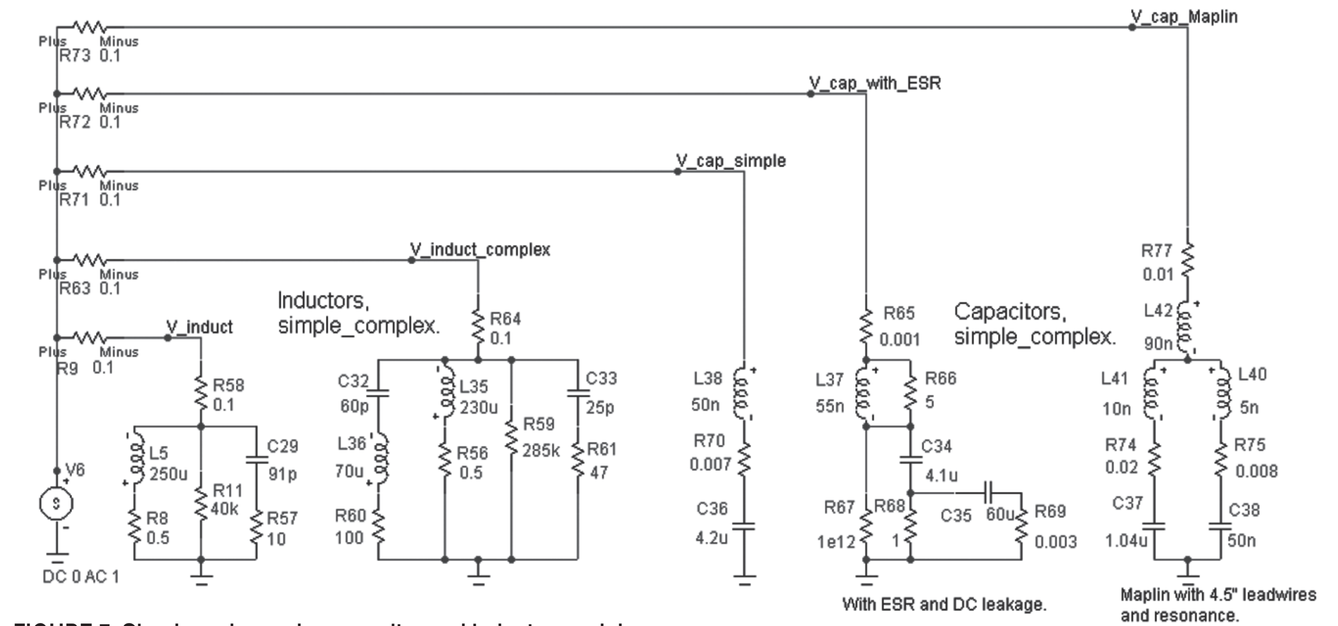

FIGURE 7: Simple and complex capacitor and inductor models.

ponent is then better described using a distributed, transmission line like, model. To complicate matters further, in real components the values of these parasitic elements invariably change with measurement frequency. SPICE allows model values to be modified using the ".model" statement for transient or frequency "F" factors for AC simulations, but these two options are mutually exclusive.

CAPACITORS

Begin by examining an improved capacitor model (**Fig. 7**). The ideal capacitor exhibits a 90° phase angle between its applied voltage and through current, but every practical capacitor must use metallic electrodes and metallic external connections, both of which introduce resistance and inductance. Any solid dielectric insulator exhibits its own dielectric losses. Combined together, these result in a phase angle at low frequency less than 90° and increasingly so with an increase in frequency. This degraded phase angle can be represented by using the appropriate, very high value shunt resistor, which, placed across a perfect capacitor, results in the same phase angle. This high resistance value is usually measured as conductance in siemens, the reciprocal of ohms.

This degraded phase angle is more usually represented by the equivalent very low value resistor in series with the perfect capacitor. This ESR value or equivalent series resistance can also be derived from measured values for capacitance and tanδ by multiplying tanδ by Xc, the capacitive reactance at that specific frequency; tanδ, of course, is frequency dependent. The capacitor electrodes and connection resistances exhibit inductance, which also acts in series with the capacitor.

A reasonable estimate for self-inductance is to allow 7.5nH for each 1cm length of straight leadout wire, plus a slightly lesser amount for each 1cm of the capacitor body length. Should the maker's impedance plot be available, then you can more accurately calculate self-inductance knowing capacitance value and the capacitor's self-resonant frequency. At self-resonance, the capacitor's reactance and self-inductance become of equal value but opposite phase angle, so they cancel out.

Hence, $L = \left(\frac{1}{4 \times \pi^2 \times F^2 \times C}\right)$

The most nearly perfect capacitor would exhibit near constant degraded phase angle or tanδ with frequency. Because for each doubling in frequency, capacitive reactance halves, the ESR for the near perfect capacitor must also halve. In a practical capacitor, ESR is frequency dependent, almost but not quite halving for each octave increase in frequency. When using the shunt resistor or G, its equivalent conductance, both values are strongly frequency dependent.

In this case, to maintain this degraded measured phase angle, the shunt resistance must more than halve and conductance (G) must more than double for each octave increase in frequency. At some frequency the metallic electrode and connection resistances dominate the dielectric contribution. ESR then reaches a minimum value, after which it slowly increases due to "skin" effects. It is certainly never a constant as many wish to believe.

The simplest fixed-value three-component capacitor model shown in **Fig. 7** can suffice over a narrow frequency range both for AC and transient simulations. Making these parameters frequency dependent extends the model useful range but negates use for transient simulations. By adding a second, larger capacitor and resistor, you can realistically model over a more useful range, both for AC and transient simulations, while retaining fixed element values. The required values are easily calculated either from makers' published graphs and data or from measured values (**Fig. 8**). Similar schematic models are provided as standard in the better, non-SPICE-based simulators.

INDUCTORS

In a similar fashion—provided the maker's impedance curves are available—you can quickly calculate the values needed for an inductor model. Unfortunately, for inductors used in crossover networks, such data usually does not exist.

You could measure the resonant frequency and impedance at resonance using a signal generator and suitable test meter, or, lacking this equipment, you could even make a vague estimate for resonant frequency based on the inductor value and the inductor plots shown. Despite not producing an accurate model, even that

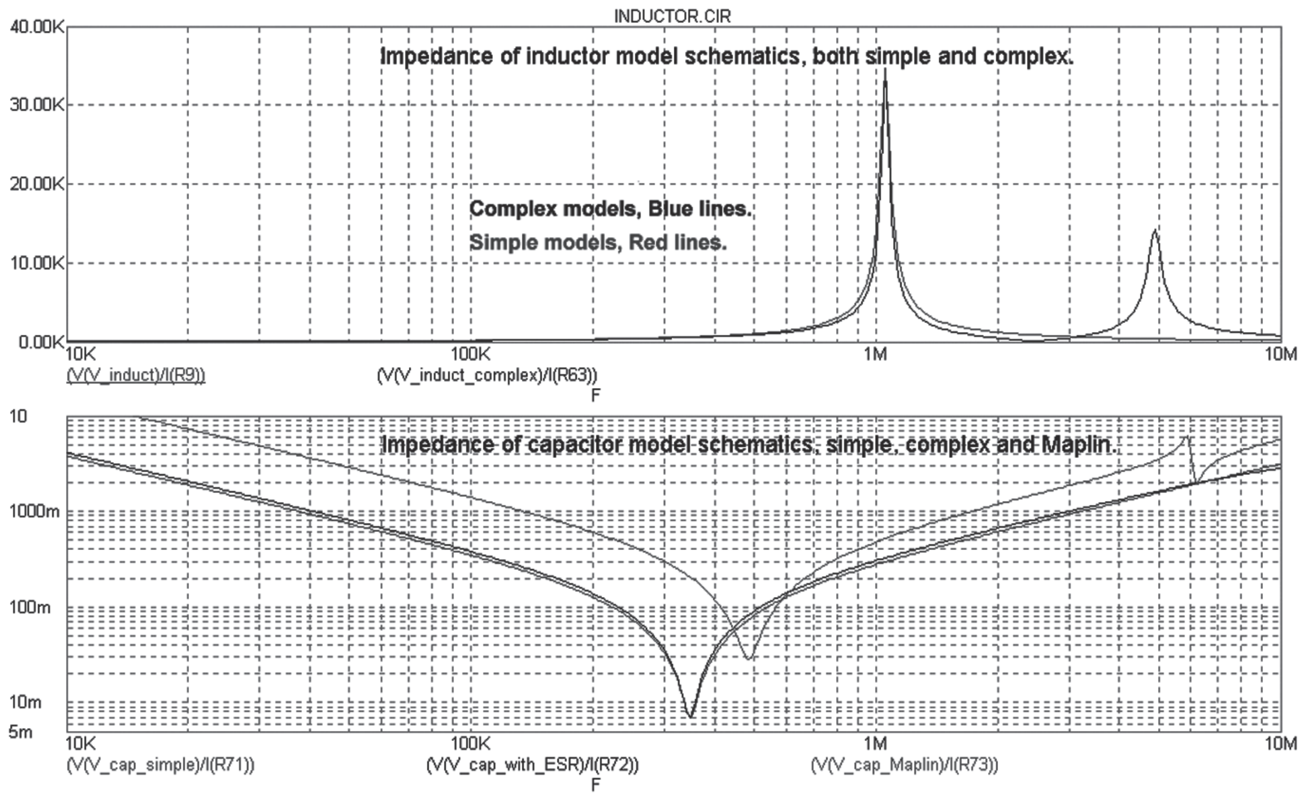

FIGURE 8: Using the schematic models in *Fig. 1*, you can quickly produce far better, more realistic simulations, closely matching actual measured values, for both capacitors and inductors.

```
$ ESP replica crossover measured Direct, impedance and phase.
$ No amplifier zobel or cable used.
$ Impedance magnitude and phase measured.
$ Frequency checked using freq counters.

.define ListVal (0,0,0)
+(1000,4.89,11.3)
+(2000,10.61,19)
+(2300,13.1,12)
+(2470,13.49,1
+(2700,13.2,-7))
+(3000,10.77,-12.6)
+(4000,5.48,-10.6)
+(5000,3.92,-0.8)
+(6000,4.01,9.3)
+(7000,4.51,15.6)
+(8000,5.20,20.4)
+(9000,5.86,24.4)
+(10000,6.58,28.8)
+(11000,7.18,31)
+(12000,7.83,33.5)
+(13000,8.41,35.2)
+(14000,8.98,36.6)
+(15000,9.56,38)
+(16000,10.11,39.6)
+(17000,10.66,40.8)
+(18000,11.20,42)
+(19000,11.75,43)
+(20000,12.31,44)
```

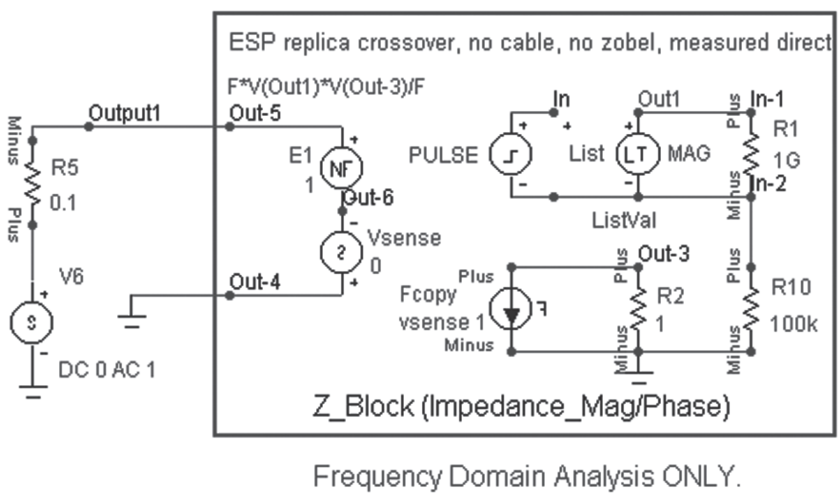

FIGURE 9: SPICE "One Port" or "Z_block" model subcircuit which you can use in simulations.

rough "guesstimate" model would be far more representative of a real inductor at high frequency than relying on the simple, basic SPICE default "perfect" inductor.

LOUDSPEAKERS

The usual published equivalent circuit for a loudspeaker driver has been devised to provide simulations only at audible frequencies. Such models then assume the speaker driver continues to act as a perfect inductor, so they do not include the inevitable self-capacitance. As a result, these models assume that speaker impedance continually increases with frequency. Unfortunately, that ever-increasing impedance is not found when measuring actual speaker drivers even at high frequency. For this article I measured impedance and phase angle of a representative range of speaker drivers, from 1kHz to 10MHz.

The T27 tweeter, used to assemble my ESP_replica circuit, was peak resonant with 825Ω at 2.4MHz, followed by a low of 105Ω at 7MHz to 390Ω at 10MHz. A Kef B110B speaker resonated with 570Ω at 865kHz, then a low of 75Ω at 3.5MHz, a small 108Ω peak at 5.85MHz to another low of 72.5Ω at 9.5MHz. I also tested 4″, 8″, and 10″ low-cost full-range drivers. All exhibited a remarkably similar series of impedance peaks followed by a number of lower impedance ripples. The bass driver used to assemble my ESP_replica network peaked with 843Ω at 500kHz, followed by multiple much smaller resonances at 1.4MHz, 2.5MHz, 3.63MHz, before finally settling to a reasonably steady 120Ω.

A Goodmans 250W 18″ "power" bass unit also exhibited multiple resonances, with 495Ω at 400kHz, 138Ω at 800kHz, 400Ω at 1.4MHz, 272Ω at 2.05MHz, 360Ω at 2.5MHz, 213Ω at 3.6MHz, and finally 160Ω at 6MHz, before settling down around 100Ω at 10MHz. While each of these plots differed, all measured speaker impedances increased up to a first major resonant peak, resulting from the inductance and self-capacitance of the voice coil and at frequencies usually between 500kHz and 1MHz, followed by a series of notably lesser impedance troughs and peaks, like those found measuring most inductors. By 10MHz all resonances had flattened to a relatively consistent, moderate impedance, typically around 100Ω. None of these drivers measured as a particularly high impedance following this first resonant peak.

The Z_block model (**Fig. 9**) allows a CSV listing of measured frequency, impedance, and phase angle parameters to be displayed on screen or used together with other components in SPICE simulation.

You may wonder why I chose to use the Z_block to represent my test inductors. Why not simply model their schematics using SPICE? At audible frequencies with modest component values, that can work quite well, however, at higher frequencies every component used—whether inductor, resistor, capacitor, and especially speaker drivers—must use complex, multicomponent models, to accurately match resonant frequencies. Every inductor or speaker voice coil includes significant self-capacitance and resonant frequency peaks and troughs. Simplistic SPICE simulation of an inductor shows impedance continually increasing with frequency, quite unlike the measured values' resonant peaks and troughs, so this can lead to false conclusions.

However, you must always remember that large value capacitors are series resonant at audio frequency. The Elna 4700μ 63V aluminum electrolytic used in my power amplifier was series resonant at 7.5kHz.

Accurately measuring an inductor, capacitor, or a complete speaker system—simply inputting measured values of impedance and phase angle by frequency into the Z_block as shown—is quicker, simpler, and most important, error free, producing the most accurate simulations possible for any component or even a complex speaker connected via its speaker cable. *aX*

Cyril Bateman's extensive work on capacitors as originally published in *Electronics World*, is now available on CD from Old Colony Sound Lab, PO Box 876, Peterborough, NH 03458, 888-924-9465, e-mail: custserv@audioXpress.com.