Simulating Inductors and networks.

Using the Micro-cap7 software, CB introduces a hands on approach to Spice circuit simulation to devise new, improved, user models, able to accurately mimic inductor behaviour 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 idealised, theoretically perfect components, suited for use with the small values used when modelling 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 analysis are made using large signal transient simulation to produce a time domain waveform of the circuit's behaviour, just like probing the circuit using an oscilloscope. When modelling in the time domain, Spice models can be modified to account for amplitude non-linearity. Small signal AC, frequency domain simulations can also be performed and when using Spice or a Spice equivalent simulator, capacitor resistor and inductor models can be modified to account also for parameter changes with frequency. However within Spice, these frequency and amplitude dependant parameters are mutually exclusive, one 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.

Unfortunately, few component makers provide realistic "Spice" models, unless their components are specifically intended for use at high frequencies, for example Coilcraft who do provide 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, results in large errors and misinterpretation of circuit behaviour. Let us examine actual measurements of a few "real" inductors.

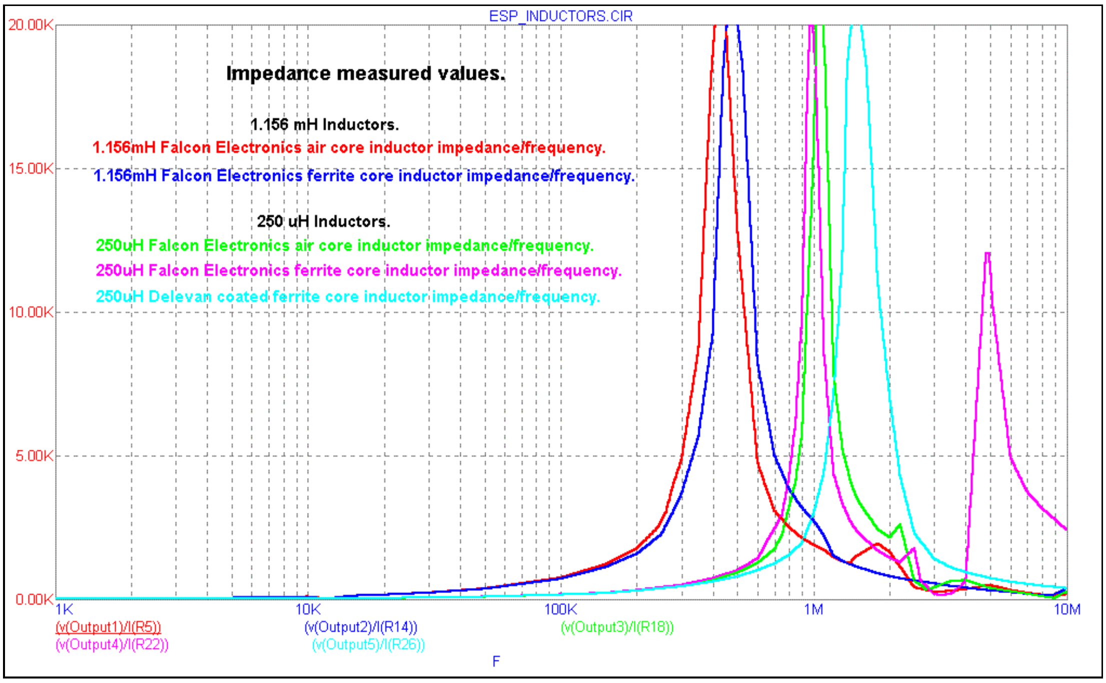


Fig.1. Measured impedance of low loss inductors, both air cored and ferrite cored, designed for use in loudspeaker crossovers.

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 care is taken to leave sufficient unwound space, typically some 90° between start and finish winds. Next we examine typical "Spice" plots using the default inductor models provided as standard.

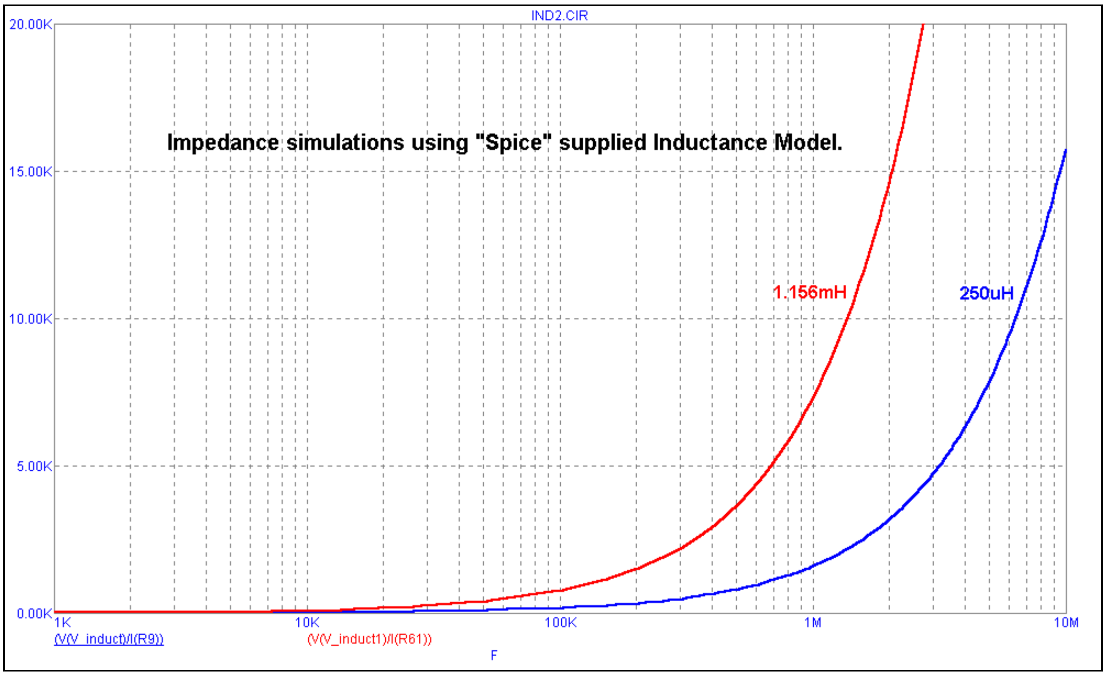


Fig.2. Simulated impedances of 1.156mH and 250µH inductance, same values as measured for figure 1, using Spice default models.

The Spice default inductor model agrees with my measured values at audible frequencies and continues to agree up to 100kHz. However by 200kHz we 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 figure 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 figure 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 a knowledge of an inductor's resonant frequency, we can easily devise a more appropriate computer model. The effective self capacitance can be calculated using the equation:- $C=\frac{1}{4\pi^2F^2L}$ derived from the more common equation $F=\frac{1}{2\pi\sqrt{LC}}$. **eqn.1**

Using the 400kHz resonance for the 1.156mH inductor, figure 1, we find it's equivalent capacitance, the combined effect of it's 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 our measured values. I find that a peak impedance shunt resistance of 40kΩ makes a good starting value, the measured DCR is series with the inductance, less say 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 ? Figure 3.

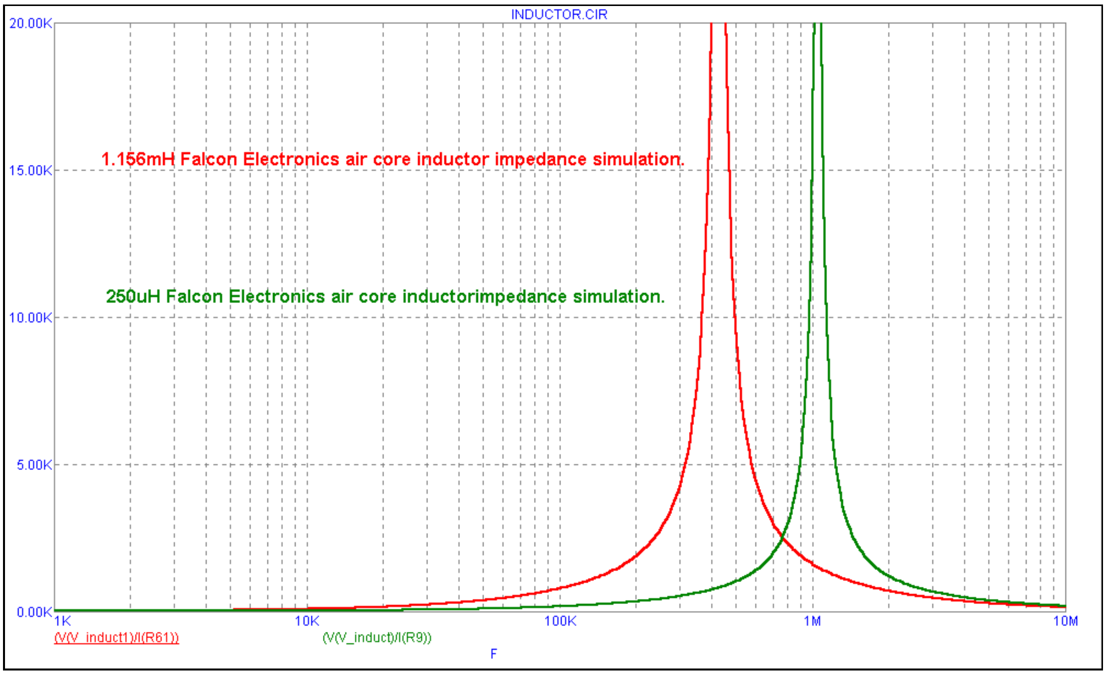


Fig.3. Simulations which closely approximate my measured impedance values for the first or main resonances of the 1.156mH also 250µH Falcon Electronics air core inductors, shown in figure 1.

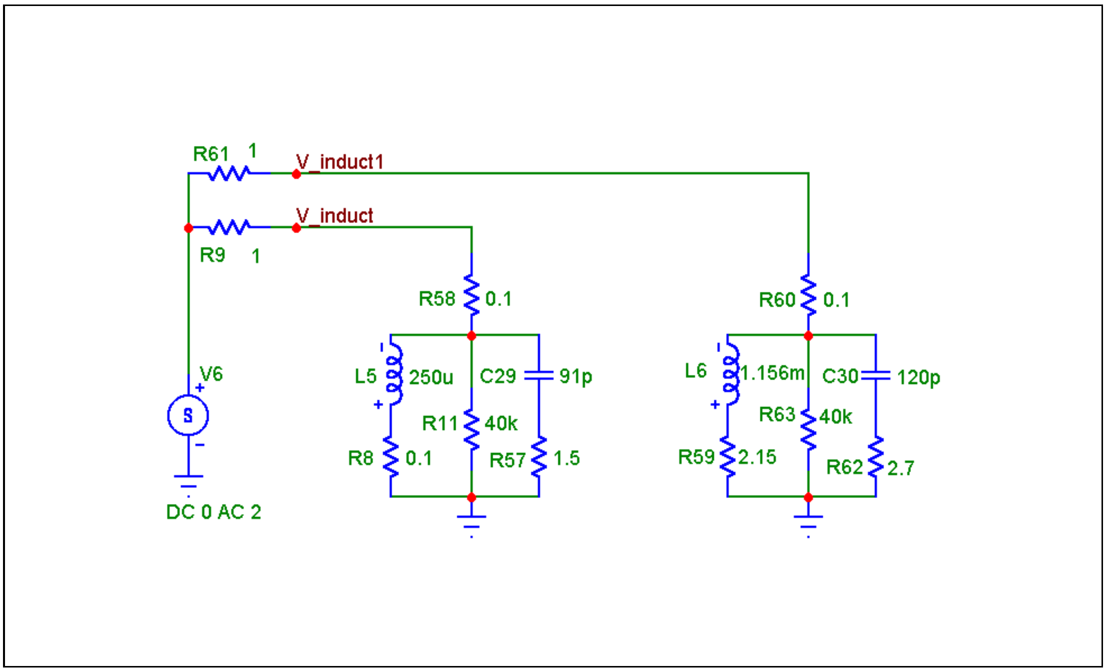


Fig.4. The simple Spice models as used for the figure 3 realistic simulations, of first or main resonances for the 250µH and 1.156mH Falcon Electronics inductors.

With a few extra components, we 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 require several (many) additional nodes. When the impedance of a driver is measured to say 10MHz, in addition to the usual audible frequency resonances which are usually well modelled, the drivers I measured ranged from a 18” 250watt Goodmans Power series with a peak impedance of 495Ω at 400kHz a Kef B110 had 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. Other drivers all measured as intermediate impedance peak values, within this frequency band. 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, 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.

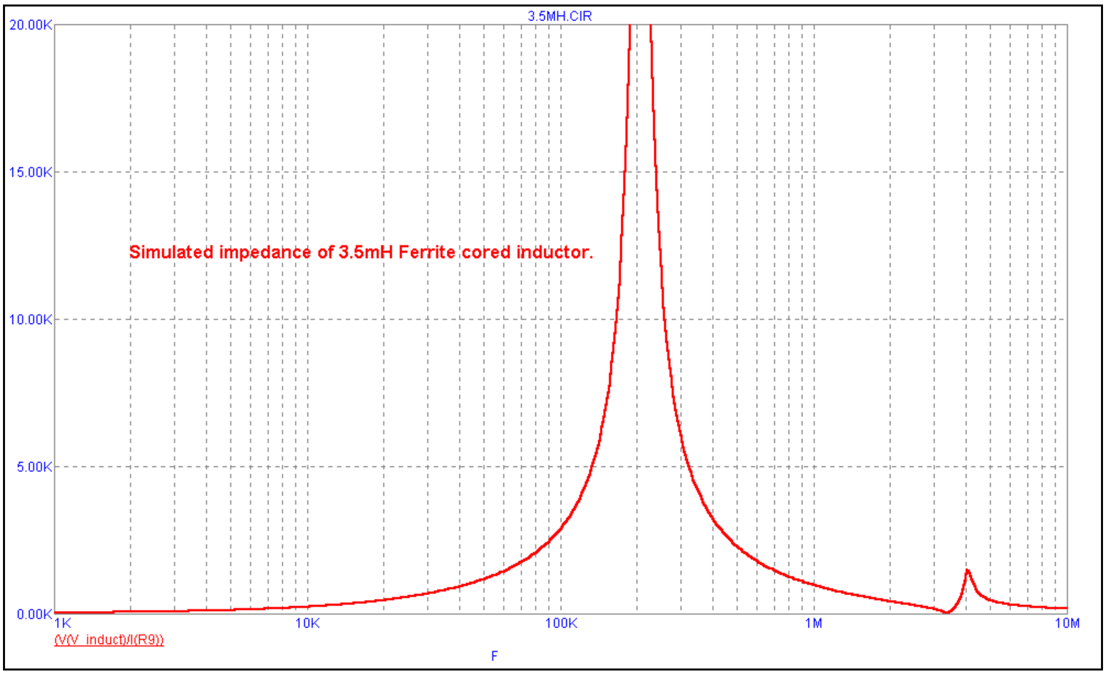


Fig.5. Simulated impedance of 3.5mH, ferrite cored inductor as used in my horn loaded speaker crossover network.

Actual measured peak impedance was 40kΩ at 210kHz.

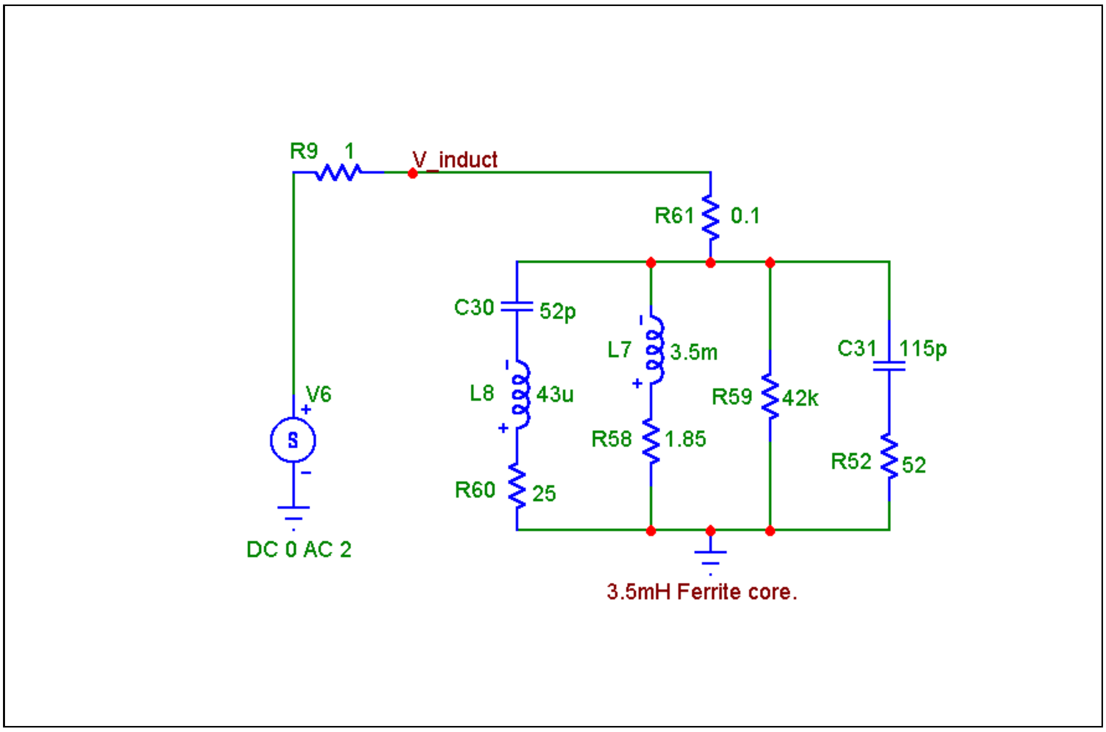


Fig.6. Model used for figure 5 simulation.

At frequencies higher than this inductive resonant peak, the inductive leading phase no longer applies, the measured phase angles now lag, 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, idealised models for capacitors, resistors, inductors and transmission lines provided in most Spice based simulators can be useful for DC, transient and low frequency AC modelling, 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 do 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 component 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.

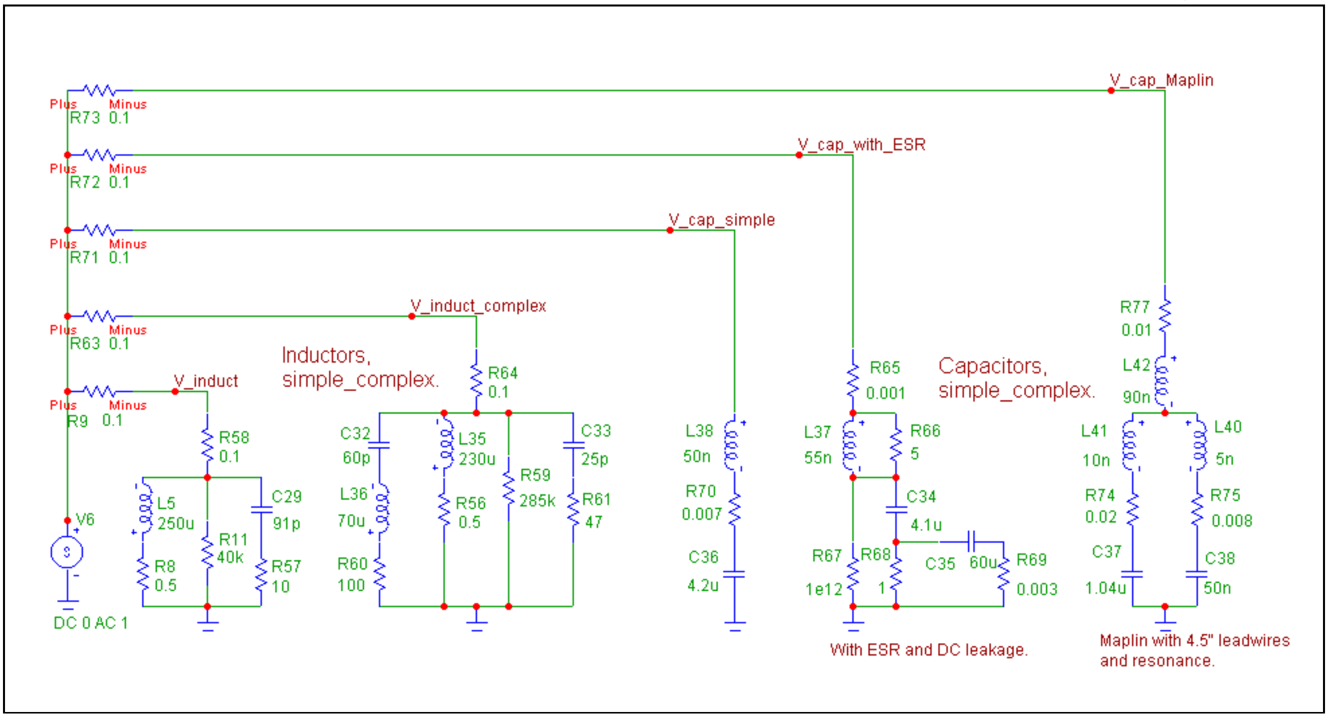
Capacitors.
We begin by examining an improved capacitor model. The ideal capacitor exhibits a 90° phase angle between its applied voltage and through current, but every practical capacitor must use metallic electrodes and metallic external connections, both introduce resistance and inductance. Any solid 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 increase in frequency. This degraded phase angle can be represented either 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 dependant. 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 makers impedance plot be available then we 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 cancel out.

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

The most nearly perfect capacitor would exhibit near constant degraded phase angle or tanδ with frequency. Since for each doubling in frequency, capacitive reactance halves, so this ESR for our near perfect capacitor must also halve. In a practical capacitor, ESR is frequency dependant, 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 dependant. 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.

Figure 7.



The simplest fixed value three component capacitor model shown in figure 1 above, can suffice over a narrow frequency range both for AC and transient simulations. Making these parameters frequency dependant, extends the model useful range but negates use for transient simulations. By adding a second, larger capacitor and resistor we 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.

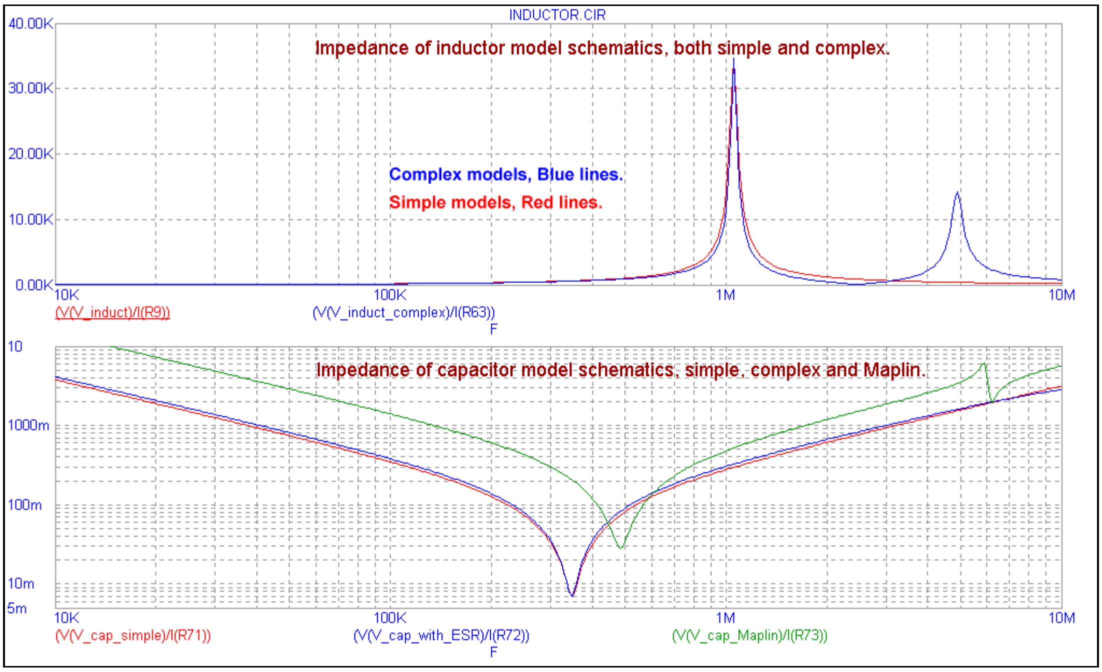


Figure 8. Using the schematic models in figure 1 we can quickly produce far better, more realistic simulations, closely matching actual measured values, for both capacitors and inductors.

Similar schematic models are provided as standard in the better, non Spice based, simulators.

Inductors.

In similar fashion, provided the makers impedance curves are available, we can quickly calculate the values needed for an inductor model. Unfortunately, for inductors used in crossover networks, such data usually does not exist. The resonant frequency and impedance at resonance could be measured using a signal generator and suitable test meter, or lacking this equipment, we could even make a vague estimate for resonant frequency based on the inductor value and the inductor plots shown. While not producing an accurate model, even that rough “guestimate” model would be far more representative of a real inductor at high frequency, than when relying on the simple, basic, Spice default “perfect” inductor.

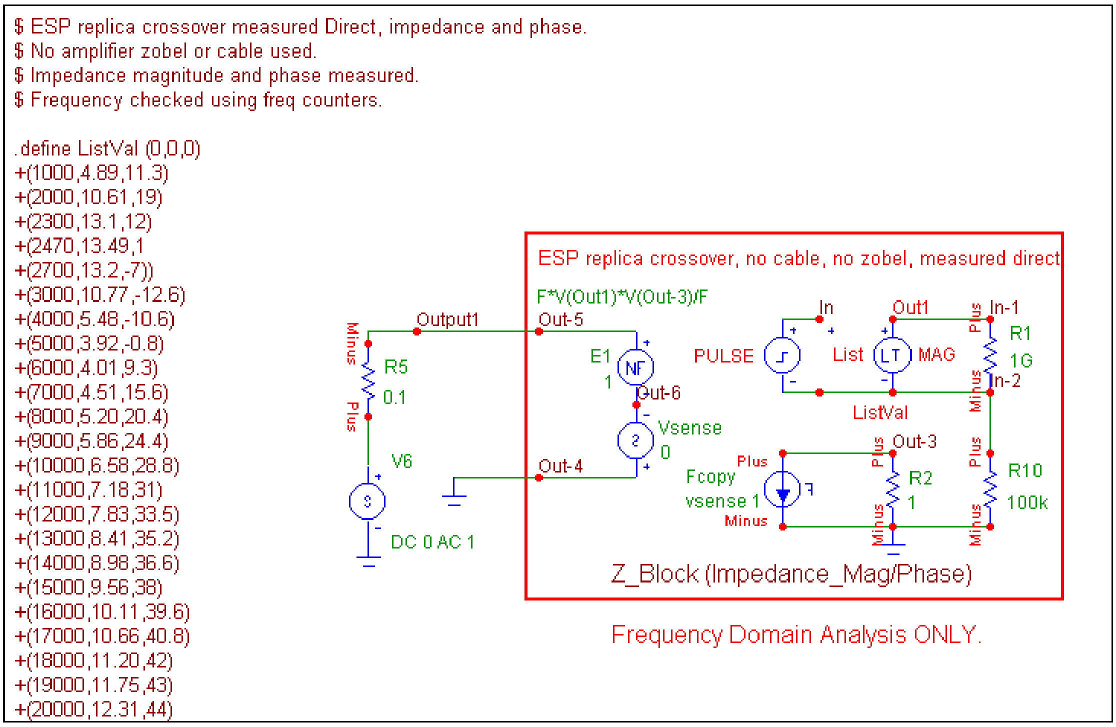
Loudspeakers.

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 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 paper 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 250 watt 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 2.5m, 213Ω at 3.6MHz, finally to 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.

Spice "One Port" or "Z_block" model subcircuit which can be used in simulations.



This Z_block model allows a CSV listing of measured frequency, impedance and phase angle parameters, see column left above, to be displayed on screen or used together with other components in Spice simulation.

You may wonder why I choose to use the above 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 so the speaker drivers, for accuracy must use complex, multicomponent models, to 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 leads to false conclusions.

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

Accurately measuring an inductor, capacitor or as in the figure 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, is error free, producing the most accurate simulations possible, for any component or even a complex speaker connected via its speaker cable.