Modelling Non-Ideal Inductors in SPICE

Martin O'Hara

Technical Manager, Newport Components, Milton Keynes

November 1994

Abstract

The non-ideal inductor exhibits both self resonance and non-linear current characteristics. These effects are modelled in SPICE by adding only 3 additional elements to model the physical inductor characteristics of DC resistance, wire capacitance and magnetic core loss. The values for these model parameters can all be obtained from standard data sheet parameters via a few simple calculations. The resulting model gives accurate impedance and phase simulations over a wide frequency range and over the peak resonance frequency. The DC current saturation characteristics is modelled by a simple second order polynomial that gives a close simulation to measured performance over twice the recommended DC current limit. Comparisons of measured inductor performance and simulation results are given to illustrate the proximity of the models to real inductor behaviour.

Modelling Non-Ideal Inductors in SPICE

Introduction

Modelling of inductors and inductive elements in SPICE has to date been of low importance to analogue circuit designers. This is partly because SPICE was developed primarily for IC design where inductive elements are usually parasitic and very small.

The widespread use of SPICE for discrete analogue circuit design has seen the program being used to analyze switching power supplies and filters in which the behaviour of the inductive element is critical to the accuracy of the simulation. In general these circuits operate using ideal inductors reasonably well since the current and frequency of operation are in the ideal operating region of the inductors used (i.e. relatively low frequency and well below the saturation current limit).

More recent applications employing inductive elements are electromagnetic interference (EMI) filters, in which the response across a very wide range of frequencies needs to be examined. Likewise employing inductors in DC power supply filters can put the inductor near its DC saturation current limit. In both later cases the modelling of the non-ideal behaviour of the inductor is important for accurate predictions of circuit performance.

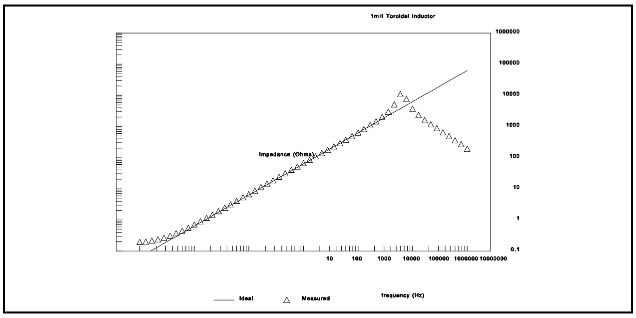
Ideal Inductor Behaviour

In an ideal inductor the impedance (Z) is purely reactive and proportional to the inductance (L) only;

Install Equation Editor and double-click here to view equation.

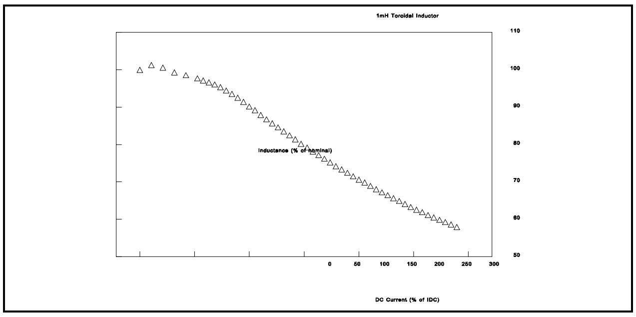
The phase of signal across the ideal inductor would always be +90° out of phase with the applied voltage and there would be no effect of DC current bias on the behaviour of an ideal inductor.

Real Inductor Behaviour



If we compare the measured frequency response for the impedance of a real inductor to the ideal model we can see two distinct differences at either end of the frequency spectrum (figure 1). At the low frequency (near DC) there is a dominant resistive element, observed in the constant impedance value and loss of the phase shift. At high frequency the inductor goes though a resonance peak and the impedance then falls and a voltage phase shift of -90° is observed, indicative of capacitive dominance. The frequency response is therefore observed to be non-ideal, however, it can be stated that near ideal behaviour does occur over the majority of the inductors operating region.

Under DC current bias there is a loss of inductance due to magnetic saturation. This is observed as a fall in inductance as the DC current through the inductor is increased (figure 2).



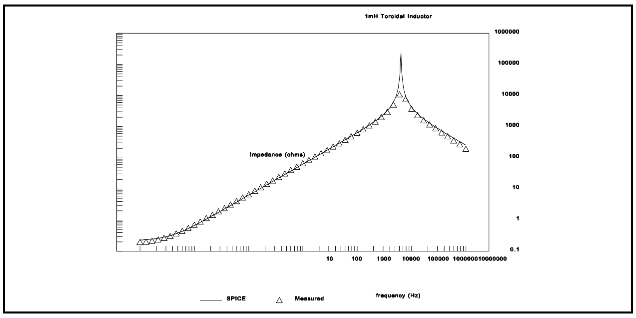
Modelling Non-Ideal Behaviour

There are two frequency dependant characteristics to model; resonance and DC resistance. Physically these are manifest in the wire wound on the core in manufacturing the inductor. The DC resistance can be considered as simply that of the wire and resonance being due to its the self capacitance. Fortunately the specification for an inductor usually includes the DC resistance value (R_{DC}) and a self resonance frequency (ω_O), hence these parameters are readily available.

The self capacitance (C_p) can be obtained from the self resonant frequency of the inductor. At the self resonant frequency the reactance of the wire capacitance (X_C) and the reactance of the inductance (X_L) are equal as they forma parallel 'tank' circuit. Hence the capacitance is given by;

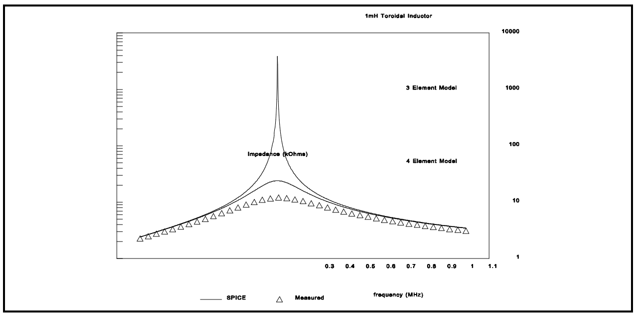
Install Equation Editor and double-click here to view equation.

The inductor can now be modelled for frequency dependence using 3 ideal elements, the inductor itself, a series resistor and a parallel capacitor. However when this is used there is still a lack of accuracy at peak resonance (figure 3).



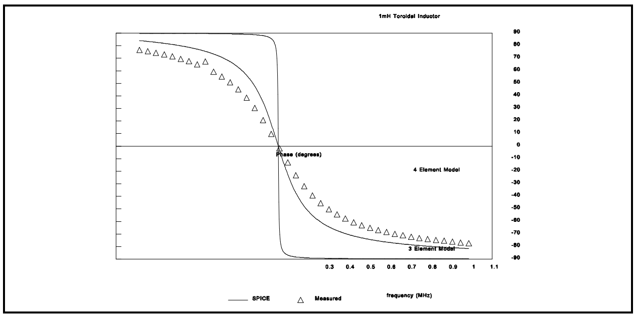
The peak resonance simulated using the 3 ideal element model has a much larger peak impedance value than the measured result (figure 4). This is due to magnetic losses in the core and is indicated in the inductor specification by a quality factor (Q) value. This loss factor occurs across the whole frequency range and is indicative of a resistive loss element, hence, can be modelled by a parallel resistor (R_P) of a large value so that it only effects overall impedance when the LC resonance peak occurs, otherwise the L and C individual reactance dominate.

The quality factor is a measure of the ratio of resistive to reactive elements of the impedance. This resistive value is only noticeable at peak resonance when the reactance tends to infinity, hence the impedance becomes limited by this resistive loss value. The value of R_P can therefore be calculated from the inductive reactance at resonance and the



Install Equation Editor and double-click here to view equation.

quality factor from the equation;
This additional resistor gives a close fit over the peak resonance for both impedance and phase curves (figure 5) while having negligible effect over the other operating regions of the inductor simulation.



An additional three components are consequently required to accurately simulate the non-ideal frequency behaviour of an inductor; a series resistor (R_{DC}), a parallel capacitor (C_P) and a parallel resistor (R_P).

DC Saturation

As the DC current flowing through an inductor is increased, the inductance falls. This effect is caused by DC saturation of the magnetic material. The shape of the fall is generally polynomial over a range of DC current, at high DC current either the inductor completely saturates and the inductance reduces to near zero, or reaches a DC saturation limit of . The high DC current saturation curve is dependant on the inductor core shape (toroid or bobbin for example), wire gauge and onset of thermal effects in the core.

In an inductor specification, usually a maximum DC current rating is given (I_{DC}), this is often the DC current at which the inductance has fallen by 10% of its original value. The figure can be used to model the fall of inductance up to this DC current limit using a polynomial equation. In version 2 of SPICE (i.e. SPICE 2G6) the polynomial inductor equation is built into the inductor model.

Install Equation Editor and double-click here to view equation.

The polynomial inductance (L) is specified by the equation;
Where $n \leq 20$ and L_0 is the nominal zero bias inductance value.
Consequently using this model and the data sheet value we can approximate the inductance DC saturation with a simple second order polynomial using the

Install Equation Editor and double-click here to view equation.

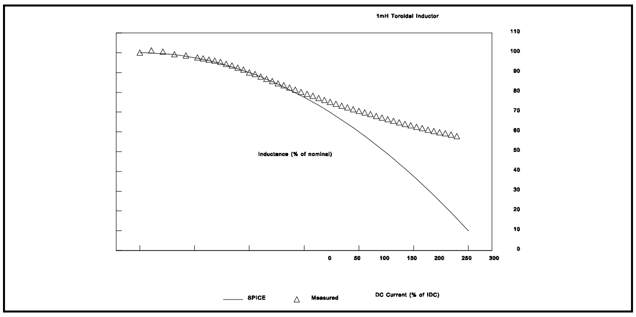
approximation;

Install Equation Editor and double-click here to view equation.

Yielding a second order coefficient of;
The first order coefficient will have to be specified as zero (L_1=0).

```
L1 1 2 POLY L0 0 L2
```

In version 3 of SPICE (i.e. SPICE 3E2 or later) the polynomial inductor is no longer available and a more complex method of modelling this effect is required. The polynomial current effect can be modelled with a zero voltage source to measure the current passing through the inductor



Install Equation Editor and double-click here to view equation.

```
.SUBCKT POLYL 1 2
V1 1 3 0
LO 3 2 Lo
B1 2 3 I=I(V1)^3*L3E2
.ENDS
```

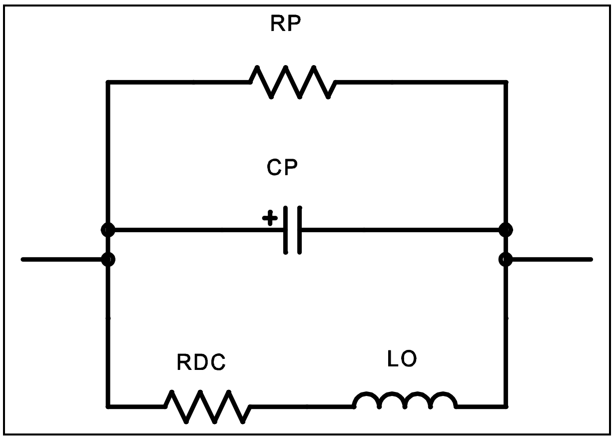
itself and a non-linear current source (B element) offsetting the applied current. The derivation of the coefficients which correlate with the SPICE 2G6 polynomial model is given in appendix A.

Essentially the coefficients for SPICE 3E2 can be calculated using the I_{DC} value again, or the L_2 figure calculated for the SPICE 2G6 polynomial model;
The simulated result for the polynomial model is close to the measured inductance curve over a range of DC current bias values up to twice the maximum DC current value (figure 6).

The limiting accuracy of the second order polynomial method is that eventually it will yield negative inductance values which are not observed in the real component under DC current bias conditions (negative values occur for DC current bias greater than $\sqrt{10}\ I_{DC}$). The main difficulty with increasing the models' ability to simulate accurately at higher DC current is the need for extra measurements and the effect of different core shapes yielding differing DC saturation curves. The higher order effect is also complicated by temperature rises in the inductor core material due to DC saturation which can also alter the curve shape. In this paper the effect of core form and DC current thermal stress will not be covered, this is being examined by the author for future work.

Summary

It is possible to simulate several complex aspects of inductor operation in SPICE using only 3 extra passive elements (figure 9) and a simple polynomial expression. The resulting model gives accurate inductor simulations over a wide range of operating conditions with a minimal increase in computation time (only one extra node is introduced). The additional model elements are all determined from available data book parameters, hence, no additional measurements are required.



References

[1] SPICE Version 2G1 User's Guide, A. Vladimirescu, A.R. Newton, D.O. Pederson, UCB, 1980 (+2G6 upgrade, undated).

[2] Electric Circuits, 2nd ed., J.A. Edminster, Schaum's Outline Series, McGraw-Hill, 1983.

[3] Soft Ferrites, 2nd ed., E.C. Snelling, Butterworths, 1988.

[4] IsSPICE 3 User's Guide, Intusoft, 1992

[5] Private communication, M. Penberth, Technology Sources Ltd, November 1993.

[6] Inductor Databook, Newport Components Limited, 1993.

SPICE 2G6 Example

The following example is a model for Newport Components' 1400 series 1mH inductor (14 105 16) where L_0=1mH, R_{DC}=0.461Ω, I_{DC}=1.6A, Q=30 and f_0=1.4MHz.

```
.SUBCKT IND14105  1  2
LO  3  2  POLY  1E-3  0  -3.91E-5
RDC 1  3  0.461
CP  1  2  12.9E-12
RP  1  2  264K
.ENDS
```

SPICE 3E2 Example

The following example is a model for a Newport Components 1400 series 1mH inductor (14 105 16) where L_0=1mH, R_{DC}=0.461Ω, I_{DC}=1.6A, Q=30 and f_0=1.4MHz.

```
.SUBCKT L14105  1  2
LO  1  4  1E-3
V1  4  3  DC 0
B1  4  1  I=I(V1)^3*-13E-3
RDC 3  2  0.461
CP  1  2  12.9E-12
RP  1  2  264K
.ENDS L14105
```