Updating Circuit Theory - Time Step Analysis

Ian Darney

1 Introduction

The classical approach to time-domain analysis of EMI is to define the spectrum of the E-field, analyse the response of the system in the frequency domain and then transform the results back into the time domain. There is a simpler way; stay in the time domain.

Even so, the approach taken by academia would be to invoke the use of Laplace Transforms or Heaviside Transforms. Again, this would lead to some heavy-going mathematics.

Such complexity can be bypassed by using Time-Step Analysis. This is similar to the use of iterative techniques used to solve simultaneous equations. Simulation programs with integrated circuit emphasis (SPICE) have used this method for decades.

But there is still a fundamental problem; SPICE software is based on nodal analysis which invokes the concept of the Equipotential Ground. That is, a conductor at which all points are at zero voltage. This can be represented as a node to which all other voltages in the system are referred. Such a concept contradicts all the lessons learned from electromagnetic theory.

This problem can be overcome by using Mesh Analysis. This relates the strength of the E-field to a voltage source in series with the victim conductor, calculates the current in each circuit loop, and defines the response of the interface circuitry at each end of the signal link.

This article provides an introduction to the technique.

2 Basic Equations

Time step analysis is similar to the use of the iterative method used in mathematics to solve non-linear equations, where each step in the process yields a more accurate value for the unknown variables. The essential difference is that each calculation defines the condition of the circuitry at a later time. By defining the size of each time step to be sufficiently small, it can be assumed that the rate of change of each current and each voltage will be linear.

The basic equations are the simplest possible:

For an inductor
$$V = L \cdot \frac{dI}{dt} \tag{1}$$

For a resistor
$$V = R \cdot I \tag{2}$$

For a capacitor
$$V = \frac{Q}{C} \tag{3}$$

Where $$Q = \sum_{0}^{t} I \cdot dt \tag{4}$$

It follows from (4) that $$I = \frac{dQ}{dt} \tag{5}$$

given that t is the elapsed time, dt is a finite increment of time, dI is a finite increment of current and dQ is a finite increment of charge.

3 Series LCR Circuit

For the circuit illustrated in Figure 1, it is assumed that the input *Vin* is a step voltage of 1 V and the objective is to determine the waveform of the current *I*. The first step in the process is to set out the voltage equation.

$$Vin = L \cdot \frac{dI}{dt} + R \cdot I + \frac{Q}{C} \tag{6}$$

The next step is to rearrange this equation to put the variable dI on the left hand side.

$$dI = \frac{dt}{L} \cdot \left(Vin - R \cdot I - \frac{Q}{C} \right) \tag{7}$$

If the initial values of I and Q are zero, then, apart from dt, the value of all the parameters on the right hand side of the equation are known. A guess-value for this parameter can be obtained by determining the time constant.

$$\sqrt{L \cdot C} = 1 \times 10^{-5}$$

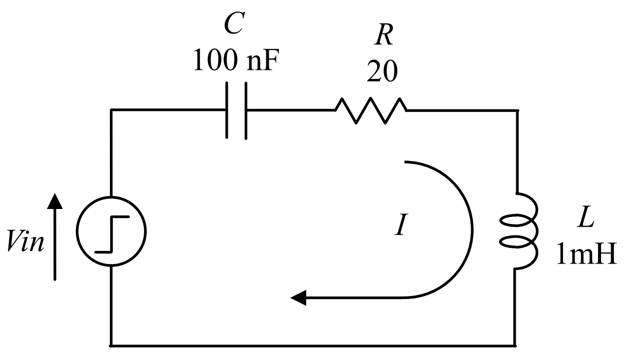

Figure 1 Series LCR Circuit

Dividing this time constant by 10 gives the guess value: $dt = 10^{-6}$.

The task then is to compile a computer program which calculates the value of dI during the time interval dt and use it to update the values of I and Q. Then these new values can be assigned to the right hand side of equation (7) and the process repeated. The value of the current I can be recorded after each time step. Figure 2 illustrates the response.

$L := 1 \cdot 10^{-3}$ $\quad C := 100 \cdot 10^{-9}$ $\quad R := 20$ $\quad \sqrt{L \cdot C} = 1 \times 10^{-5}$

$Vin := 1$ $\quad dt := 10^{-6}$ $\quad N := 100$

$I := 0$ $\quad Q := 0$

$$next(I, Q) := \left| \begin{array}{l} dI \leftarrow \frac{dt}{L} \cdot \left(Vin - R \cdot I - \frac{Q}{C} \right) \\ I \leftarrow I + dI \\ Q \leftarrow Q + I \cdot dt \\ (I \quad Q) \end{array} \right.$$

$$Iout := \left| \begin{array}{l} \text{for } i \in 1..N \\ \quad \left| \begin{array}{l} Imon_i \leftarrow I \\ (I \quad Q) \leftarrow next(I, Q) \end{array} \right. \\ Imon \end{array} \right.$$

$i := 1..N$ $\quad t_i := (i - 1) \cdot dt$

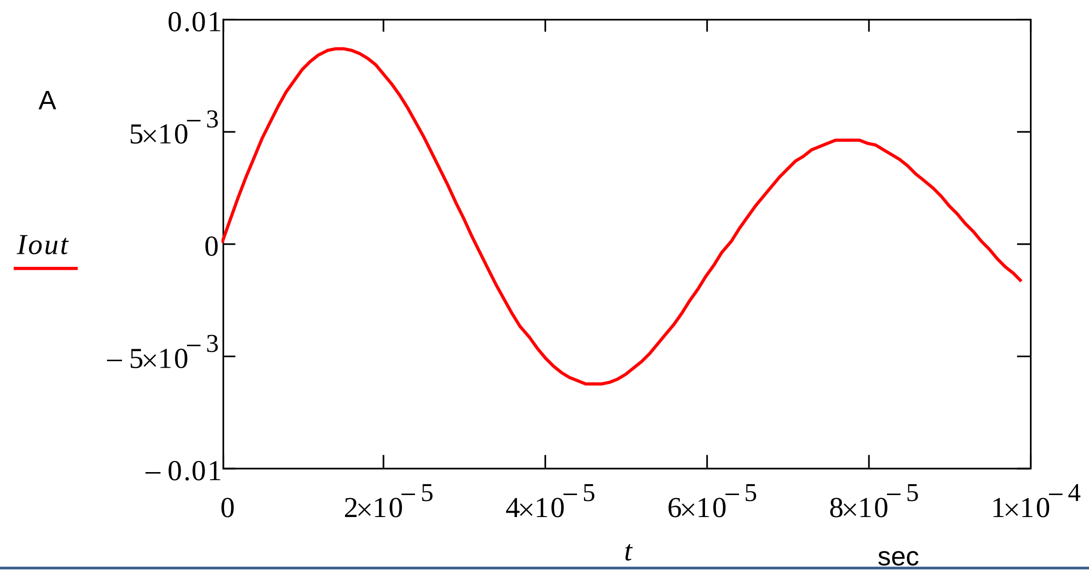

Figure 2 Transient response of series LCR circuit

4 Low Pass Filter

A slightly more complex example is that of Figure 3, which simulates a low-pass filter. To enable the model to simulate steady-state conditions, it is necessary to place a high value of resistance in parallel with the capacitance.

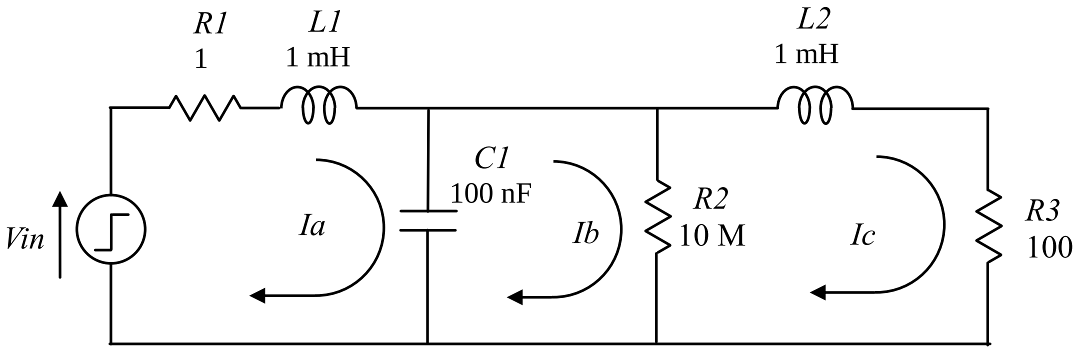

Figure 3 Low pass filter

The loop equations for this filter are

$$\begin{aligned} Vin &= R1 \cdot Ia + L1 \cdot \frac{dIa}{dt} + \frac{Ia \cdot dt}{C1} - \frac{Ib \cdot dt}{C1} \\ 0 &= -\frac{Ia \cdot dt}{C1} + \frac{Ib \cdot dt}{C1} + R2 \cdot Ib - R2 \cdot Ic \\ 0 &= -R2 \cdot Ib + R2 \cdot Ic + L2 \cdot \frac{dIc}{dt} + R3 \cdot Ic \end{aligned} \tag{8}$$

Let $$Q = (Ia - Ib) \cdot dt \tag{9}$$

Re-arranging equation (8) and invoking (9)

$$\begin{aligned} dIa &= \frac{dt}{L1} \cdot \left[Vin - R1 \cdot Ia - \frac{Q}{C1} \right] \\ Ib &= \frac{1}{R2} \cdot \left[\frac{Q}{C1} + R2 \cdot Ic \right] \\ dIc &= \frac{dt}{L2} \cdot \left[R2 \cdot Ib - R2 \cdot Ic - R3 \cdot Ic \right] \end{aligned} \tag{10}$$

The input variables of the Mathcad Worksheet are derived from the circuit model of Figure 3. The function *next* (*Ia, Ib, Ic, Q*) incorporates the set of equations defined by (10). The main program calculates a set of values for the output voltage *Vout* across 100 ohm load *R3*.

The graph at the bottom of the worksheet illustrates the response of this particular low pass filter to a 1 Volt step input.

$L1 := 1 \cdot 10^{-3}$ $\quad L2 := 1 \cdot 10^{-3}$ $\quad C1 := 100 \cdot 10^{-9}$

$R1 := 1$ $\quad R2 := 10^{7}$ $\quad R3 := 100$

$Vin := 1$ $\quad dt := 10^{-6}$ $\quad N := 200$

$Ia := 0$ $\quad Ib := 0$ $\quad Ic := 0$ $\quad Q := 0$

$$next(Ia, Ib, Ic, Q) := \left| \begin{array}{l} dIa \leftarrow \frac{dt}{L1} \cdot \left(Vin - R1 \cdot Ia - \frac{Q}{C1} \right) \\ Ib \leftarrow \frac{Q}{R2 \cdot C1} + Ic \\ dIc \leftarrow \frac{dt}{L2} \cdot [R2 \cdot Ib - (R2 + R3) \cdot Ic] \\ Ia \leftarrow Ia + dIa \\ Ic \leftarrow Ic + dIc \\ Q \leftarrow Q + (Ia - Ib) \cdot dt \\ (Ia \quad Ib \quad Ic \quad Q) \end{array} \right.$$

$$Vout := \left| \begin{array}{l} \text{for } i \in 1..N \\ \quad \left| \begin{array}{l} Vout_i \leftarrow R3 \cdot Ic \\ (Ia \quad Ib \quad Ic \quad Q) \leftarrow next(Ia, Ib, Ic, Q) \end{array} \right. \\ Vout \end{array} \right.$$

$i := 1..N$ $\qquad t_i := (i-1) \cdot dt$

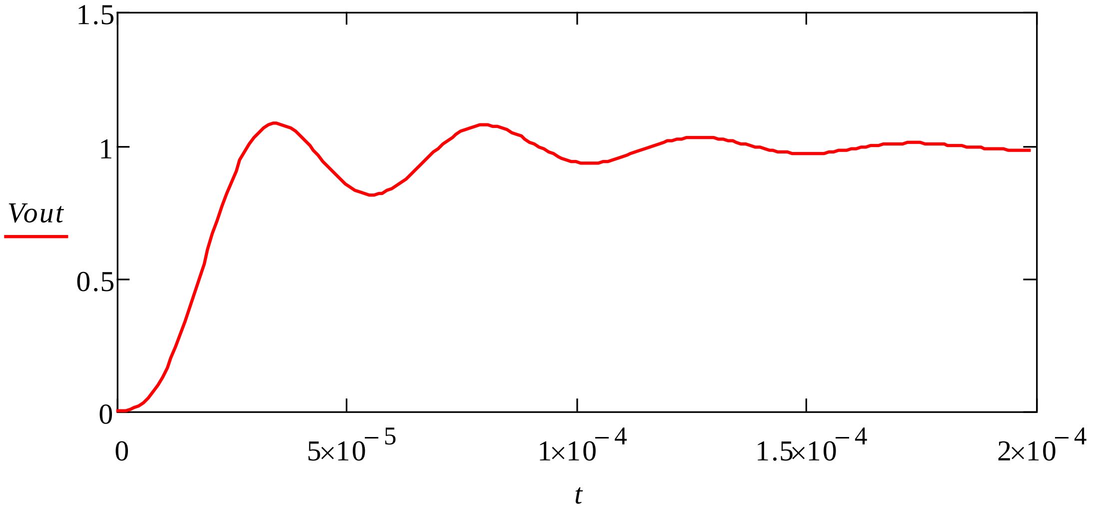

Figure 4 Transient response of low-pass filter

5 Assessment

A review of the calculations reveals that the parameters are time, voltage, charge, current, resistance, inductance and capacitance. The concepts of frequency, impedance, admittance, and phase angle have no existence in any analysis in the Time Domain.

The process used to create and run the program can be summarised:-

Define the values of the components of the network.

Define the loop equations, the value of the time step, and the waveform of the voltage source.

To calculate the values of the parameters after each time-step:-

Define the new value of the voltage source.

Define the step change of current in each inductor.

Define the new current in each resistor.

Define the new charge in each capacitor.

Provide the new values of the currents and charges as the output of the function.

To create the output waveform:-

Define the timescale.

Define the current or voltage in the selected component.

Provide the value of the selected parameter as the output after each time-step.

When the program is run, check that the output is plausible. If necessary, reduce the value of the time-step. If the process takes too long, increase the time-step.

It usually takes less than a second for the software to carry out the computations. This means that the designer can alter the value of any component and note how this affects the output waveform.

6 Conclusion

A method of using mesh analysis to compile programs to simulate the transient behaviour of circuit models has been described, and illustrated using two simple examples.

The calculations used in the above two examples can be carried out by any SPICE program. With such software, the engineer must rely on its infallibility.

The advantages of mesh analysis are that it can be developed to simulate all the mechanisms involved in the propagation of transient interference, and that the designer remains in complete control of the process.