Simulation # 2

Calculation of Power and Energy For passive components by Graphical Method

Electrical Network Simulation Laboratory (EE2701)
Department of Electrical Engineering,
NIT Rourkela

Aim of the Expt.:

- Understand the graphical significance of differentiation and Integration
- Understand piecewise signal analysis of continuous signals
- Calculate instantaneous power and energy for an electric circuit containing passive elements using Multisim simulation

Theory:

Piecewise linear functions:

A piecewise linear function is one composed of straight-line segments. In the case of differentiation, only continuous functions are considered. A continuous function is the one that does not possess abrupt jumps; all changes in the function take some amount of time to occur. The examples of continuous and discrete functions are as shown in Figure 1.

Piecewise linear functions are of importance because of two basic reasons: First, many common waveforms arising in electrical circuit applications are of this general type. Second, working with piecewise linear functions in basic transient analysis provide some insight into the physical nature of the operations that would be difficult with other functions.

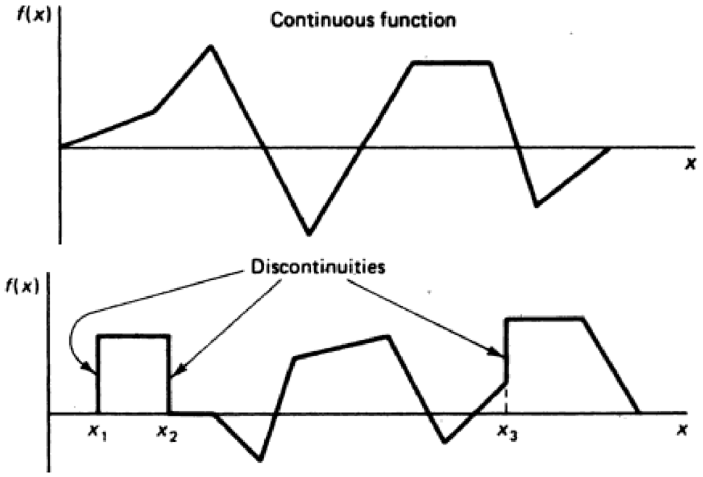

Figure 1 Examples of continuous and discrete signals

Power and Energy stored in Capacitance:

The power in the capacitor is the product of instantaneous voltage across and instantaneous current through the capacitor. The energy stored in a capacitor is a function of the voltage across it. In a capacitor at any time t, energy is given by

$$w_c(t) = \frac{1}{2}\,Cv^2(t)$$

Energy can also be obtained by integrating the power over the time period. In an ideal capacitor left open-circuited, the voltage and energy would remain stored forever. However, in practical capacitors, always have some leakage dielectric, which will eventually discharge the voltage and dissipate the energy. The energy stored in a capacitor represents potential energy since it is a function of charge at rest.

Example:

A 0.5 uF capacitor is supplied with a voltage having waveform v(t) as shown in Figure 2 determine and plot as functions of time

a) Current
b) Power
c) energy

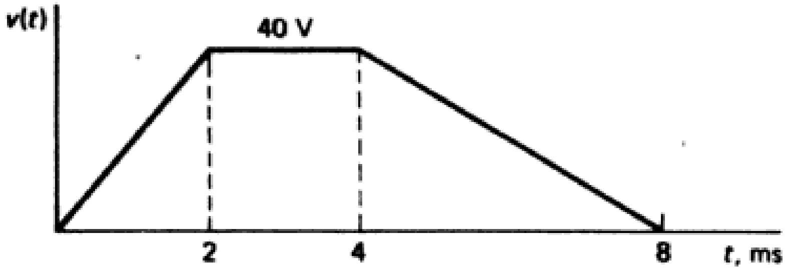

Figure 2 Piecewise linear voltage applied to capacitor

Simulation:

1. Build a circuit where a capacitor is excited by piecewise linear voltage signal as shown in Figure 3. Initial conditions of the capacitor are assumed to be zero.

2. Piecewise linear voltage source is placed under "signal voltage sources". After placing it on the workspace, double click the symbol to edit the signal. Enter the values of Time and Voltage as per the signal is shown in Figure 2. First and last voltage values are set to zero.

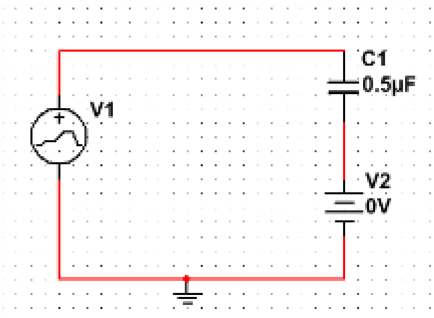

Figure 3 Circuit for plotting current waveform

3. Plot the voltage and current waveforms separately using transient analysis tab with appropriate time parameters.

4. To plot power and energy graphs, use elements like "multiplier and voltage_integrator" from the control function box as shown in Figure 4. As multiplier in Multisim uses two voltage sources as input, a current-controlled voltage source with the trans-resistance value set to 1 is used to convert current into voltage.

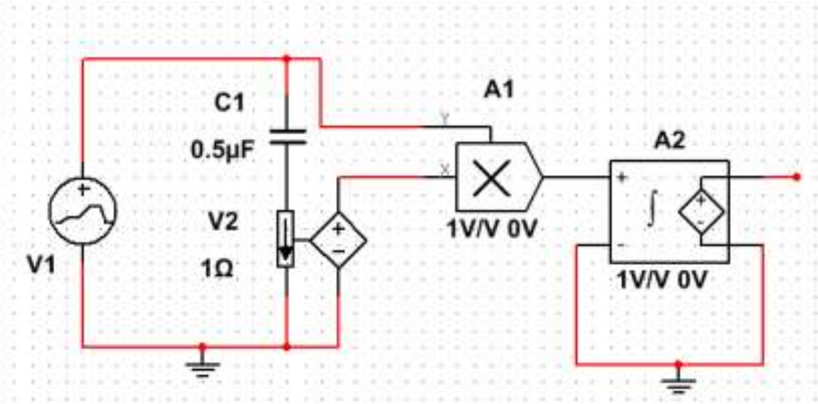

Figure 4 Circuit for plotting power and energy waveform

Questions:

1. Why a voltage source with magnitude "0" is placed in series with a capacitor in Figure 3?

2. Find the output voltage of a 0.2 uF capacitor when excited by a current source with piecewise linear current input as shown in Figure 5.

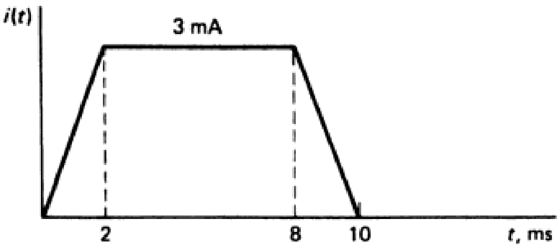

Figure 5

3. A square wave oscillates between +10V and -10 V with a frequency of 1 kHz is applied across 1-mH inductor as shown in Figure 6. The waveform starts with a positive half cycle. Plot the waveforms of voltage and current over one cycle using the current probe and oscilloscope. Trace the waveform over multiple cycles.

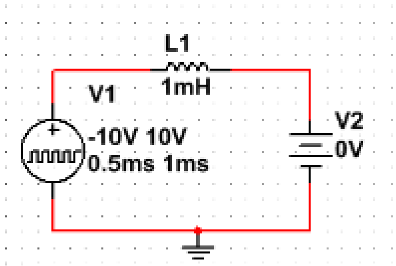

Figure 6