A SWITCHING TRANSIENTS OF UNLOADED TRANSFORMER IN POWER SYSTEM

Thesis presented in partial fulfilment for the award of the
Advance Diploma in Electrical Engineering of
INSTITUT TEKNOLOGI MARA

MOHD AZMI BIN ZAINUDDIN
Department of Electrical Engineering
INSTITUT TEKNOLOGI MARA
40450 Shah Alam, Malaysia
DECEMBER 1995

ABSTRACT

This thesis is to studies the switching transients of unloaded transformer in power system during switching operation and PSPICE software was used to determined the voltage at the unloaded(stub) cable.

Switching over voltage frequently occurs in electrical power system such switching overvoltage can damage many expensive electrical items such as transformers, cables, motors and others. The effect of overvoltage comes from a sudden increase in magnitude of the travelling wave propagation in a conductor such as transmission line. Damages to power transformers are unwelcome since continuity in power delivery may be seriously disrupted. Furthermore, repairs or replacement, are expensive and time consuming.

An example of damage to a transformers is the switching on of unloaded transformers. When the frequencies of the line and transformer matched, very high voltage may appear at the secondary terminals of the transformer. This can damage the insulation of the transformer winding, and finally lead to a flashover from winding to core.

ACKNOWLEDGEMENT

In the name of Allah s.w.t, The most Gracious who has given me the strength and ability to complete this project and report.

All perfect praises belong to Allah s.w.t, Lord of the universe. May His blessing upon the prophet Muhammad s.a.w and members of his family and companions.

I would like to express my gratitude to my project supervisor, IR. Amir Mohd Saad for his guidance and ideas in advising and assisting my project. My gratitude also to Dr. J.N. Sinha and Pn. Zuhaina for their guidance and in sharing their knowledge to made this report. Also, thank you to my classmate and friends for their suggestion and contribution to finished this project.

Mohd. Azmi Bin Zainuddin

INSTITUTE TEKNOLOGI MARA

Shah Alam

Selangor

A SWITCHING TRANSIENTS OF UNLOADED TRANSFORMER IN POWER SYSTEM

CONTENTS **PageNo**

CHAPTER 1

CHAPTER 2

CHAPTER 1

1. INTRODUCTION

This project is to study and investigate the switching overvoltage during energizing of an unloaded transformer(cable) by a circuit breaker. Furthermore, a computer simulations package PSpice(Simulated Program with Integrated Circuit Emphasis) were used to simulate the phenomenon of the transient that occurs during switching operations. The travelling wave concept is used to described the phenomenon and applied to the system.

Normally, when an unloaded power transformer is switched on via a relatively long cable, sometime extreme high voltages appear at the secondary side of the transformer[1]. These overvoltage are caused by a resonant phenomenon that occurs when the resonant frequencies of the transformer and the cable match. The resonant frequency of the cable feeder is equal to the reciprocal to four times its travel time[2].

The first approach in this project is to simulate the transient response in the basic circuit of the RC circuit, RL and RLC circuit. The second approach is to study the switching overvoltage of unloaded transformer. PSpice simulation is used to simulate the model of the system.

CHAPTER 2

2. THEORY

There are a lot of things of the basic notions concerning the transient behaviour of the circuit especially the circuit will be used to model the system in this project. All this are important to study to give the knowledge and an idea to proposed in this project.

2.1 Fundamental Notions about Electrical Transients

An electrical transient is a sudden change in circuit conditions, when a switch opens or closes or a fault occurs on a system. The transient period is usually very short. The fraction of their operating time that most circuits spend in the transient condition is insignificant compared with the time spent in the steady state.[6]

This transient periods are extremely important, for it is at such times that the circuit components are subjected to the greatest stresses from excessive currents or voltages. In extreme cases damage results. This may disable a machine, shut down a plant, or black out a city, depending upon the circuit involved. For this reason a clear appreciation of evens talking place during transient periods is essential for a full understanding of the behaviour of electric circuits.

2.2 Electrical Surges And Transients

Phenomena which are not simple periodic functions of time and which usually exist for only a short time, are referred to as natural responses of transient. When connecting or disconnecting apparatus from a supply, a disturbance occurs in the circuit current, which dies out after a short period. These transient current or voltages are associated with changes in the stored energy in inductors and capacitors. Since there is no stored energy associated with resistance, there is no transient current in a pure resistor circuit.

2.3 Starting Transients

Transient condition, particularly starting transient, can cause higher flux densities than exist in the steady-state condition. Power transformer, for reasons of economy, are not designed to absorb these transient without saturating. This condition contributes to what is usually called the "inrush current". The magnitude of the inrush current depends upon many factors including circuit characteristics.[3]

Some types of loads have starting currents much larger than operating current. Motors and the filaments of large transmitting tubes are common examples. The point on the sine wave at which switch contact is made determines the volt-time area during the first half-cycle.

If contact is made when the voltage is zero, the volt-time area during the first half-cycle will be twice that developed when switch contact is made when the voltage is at its peak value. In the former case, saturation on the half-cycle is assured. Saturation of the core reduces the value of the shunt inductance to zero. Current is then limited only by the leakage inductance and primary winding resistance. These two parameters combined constitute the short-circuit impedance of the transformer.

Core saturation effects last only a few cycles of power frequency. The inrush current will be greater if the transformer is operating at the threshold of saturation in steady state. The results of this are sometimes observed in the form of occasional blown fuses or popped circuit breakers at turn on. A small reduction in initial cost is achieved if this situation is tolerated.

It is sometimes advantages to deliberately build a large short-circuit impedance into the transformer to limit high starting currents in the load. Winding resistance can be increased for this purpose up to the point where the limitation is the insulation temperature. Increasing the leakage inductance is a better way to increase the short-circuit impedance since no additional energy is consumer dissipated in the form of heat in the transformer.

Increases the short-circuit impedance increases the regulation. If the steady-state current is predictable and constant, the poorer regulation can be tolerated. Deviations from nominal full-load voltage will be greater when the short-circuit impedance is made larger because of the limitation on the precision with which regulation can be predicted in design and controlled during manufacture.

Transformer contribute to the worsening of an inductive power factor of the load on a distribution system. Consideration of this effect is seldom given priority in equipment transformer design. The power factor will be improved when transformers are fully loaded at unity power factor and when the transformers are of minimum size for the application, since the inductive exciting current under these conditions is a smaller percentage of the total primary current. Power factor is a consideration not only in large distribution system but also in smaller system in vehicles and aircraft where keeping the size and weight of equipment to a minimum is a primary requirement.

There are several possible ways in which these abnormal disturbances can come about, but they have one thing in common. They all involve the trapping of energy somewhere in the circuit and its subsequent release. This could be due to charge on a capacitor or line/or current in an inductor. It follows that if a circuit is completely quiescent when a transient is initiated, the transient will be a normal transient. Of course it could be that this transient stores energy in the system so that subsequently when a second transient is initiated, it is abnormal.

2.4 Normal And Abnormal Switching Transients

What is considered normal or abnormal in switching transients is somewhat arbitrary. There is certainly no generally accepted definition. We have seen how, when the switch opens in a single phase circuit, it is theoretically possible for the transient recovery voltage to reach twice the peak magnitude of the system voltage. We have also observed that a discharge capacitor can attain a similar voltage if energized at the peak of the system voltage cycle and that a power frequency current can attain twice its normal peak rms value, depending on the instant at which the circuit is energized. It is reasonable to call such events normal switching transients. There are, however, other circumstances in which voltages and currents far in excess of these values can arise. Such transients, by our definition, are abnormal.

2.5 Switching Surge

Switching operations that cause overvoltages are mainly due to trapped energy in part of the circuit and the subsequent release of that energy. System configurations that are more likely the cause of overvoltages are reviewed. For example in transformer and Cable Unit switching overvoltage.

Overvoltage due to resonance may result when a lightly loaded or unloaded transformer with a long cable connection is a remotely switched system configuration. The magnitude of overvoltage may or may not exceed two times the system voltage, however surge arrested thermal capability needs to be checked as the voltage may sustain for a longer period of time.

2.6 Models For Cable

Cables can be divided into two broad categories, shielded and unshielded. The effect of the shield is to confine the electric field between itself and the conductor. Since the current almost always returns through the conductors of the other phases, the magnetic field is not confine in the same way, it is nevertheless affected by the presence of the shield because of eddy currents induced therein.

Cable around power plants are frequently layed in trays of expanded metal. The tray is normally grounded and like the shield of a shielded cable the tray acts as an equipotential surface for the electric field in unshielded cables. The sides of the tray may have substantial currents induced in them by currents in the conductor proper.

As the overhead transmission lines, there are circumstances in which a cable can be represented by a resistor equal to its surge impedance. Cable surge impedance tend to be significantly lower than surge impedance for overhead lines because the closer proximity of their conductors increases the capacitance and reduces the inductance. Values in the range 20-50 are typical. Capacitance is also enhanced by the permittivity of the electric which also reduces the speed of wave propagation along the cables. This is typically found to be one half to two thirds of the speed of light.

2.7 Representation of Transmission Lines

The voltage and current waves travel along a transmission line with the same velocity. The function of transmission lines is to transfer electrical power from one point to another point when a switch is closed, a voltage appears immediately at the input of the transmission line. However, it cannot appear instantaneously at the other points along the line, since that would require a sudden change in voltage on all the capacitances. The example of transmission line can be represented as Fig.1.

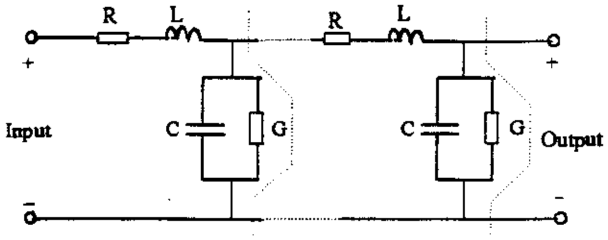


Fig.1 : Model of Transmission Line

Furthermore, since it is current that delivers the electric charge to the capacitances a sudden increase in current through the inductance would also be necessary. Since inductance opposes a current change and capacitance opposes a voltage change, the voltage and current require a finite time to propagate along the transmission line. The propagation process can be described in the following manner. When the switch is closed, the first inductance generates a back emf. in according with Lenz's Law, to initially oppose an increase in current.

The current flows through the inductor and charge the first shunt capacitance to voltage, V. This charges the next capacitor and process continues down the line. From this argument, it is apparent that voltage creates current and vice versa, thus requiring that they travel together along the transmission line. Travelling waves along the transmission when reached the load will produce reflection waves.

The voltage and current that appears at the other end, when the line is energized. The lines and cables can be represented as network resistance, inductance and capacitance. The manner in which the line and cable are represented depends on their length and the accuracy required.

2.8 Unloaded transformer

Transformer is a device that transfer energy from one system to another. A transformer can accept energy at one voltage and deliver it at another voltage. This permits electrical energy to be generated at relatively low voltages, transmitted at high voltages and low current, thus reducing line losses, and used at safe voltages. Unloaded transformer is a transformer without any loaded or we also called the "ideal" transformer. In ideal transformer would have winding with zero resistance and a losses, infinite permeability core. The efficiency would be 100 percent. Infinite permeability would result in zero exciting current and no leakage flux.

2.9 Simulation

The software being used to simulate the voltage surge occurs in this system is PSpice, 'Simulated Program with Integrated Circuit Emphasis'. The system which is represented is applied in this software with the equivalent parameters that available in the library of this software.[5]

The PSpice circuit simulation program is standard to used in Industry today. The major advantage is using PSpice in power electronics is that, with the same software, analysis and design through different levels can be done. However, to allow higher levels of simulation, simplified models for switch must be implemented in order to minimize convergence problems and reduce the run times.

CHAPTER 3

3. TRANSIENTS RESPONSE IN RL CIRCUIT

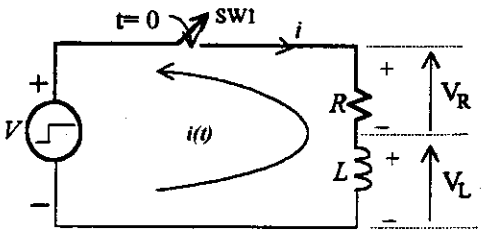


Fig. 2 : Series RL circuit with dc PWL source

Consider the RL circuit as shown in Fig.2. This circuit consists of resistance, *R* and inductance, *L*. When SW1 is closed, the current,*i* through the inductor build up from zero, establishing a voltage drop across the resistor and corresponding drop the voltage in inductor. The current will continued to increase until the voltage across the inductor drops to zero. An inductor tends to resist a change in current.

3.1 Representation of the Circuit

Initially, inductor current is zero and builds to maximum of E/R. The current is given by ;

$$i_L = \frac{E}{R}[1 - \varepsilon^{-t/\tau}] \tag{3.1}$$

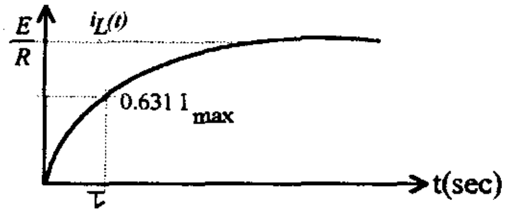


Fig.3 : Series Circuit Current

The current flow in this circuit can be represented in Fig. 3, for the RL circuit, the time constant,τ is;

$$\tau = \frac{L}{R} \tag{3.2}$$

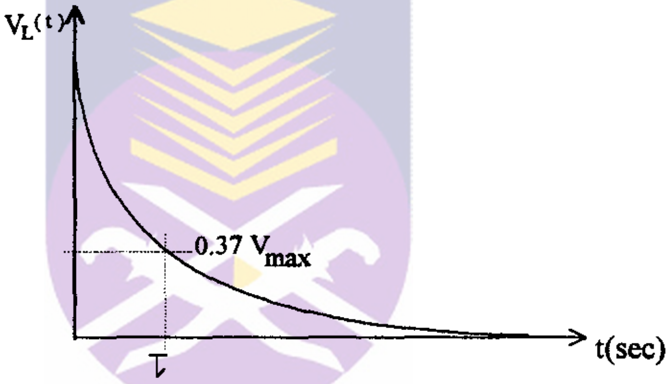


Fig.4 : Time Inductor Voltage for RL Circuit

The inductor looks initially like an open circuit, and the inductor voltage drop from E to zero volts by the expression ;

$$V_L(t) = E\varepsilon^{-t/\tau} = E\varepsilon^{-t(R/L)} \tag{3.3}$$

comparing Fig. 4 with $V_L(t)$, at $t = 0$, $V_L(t) = E\varepsilon^0 = E$, which agrees with the diagram. At $t = \infty$, $V_L(t) = E\varepsilon^{-\infty} = 0$, which also agrees with the diagram. From the basic RL circuit, we can determined the time constant by using the calculation and by simulation using PSpice.

3.2 Simulation

In this circuit, we used R= 10kΩ and L= 10mH. So, from calculation, the time constant is given as;

$$\tau = \frac{L}{R} = \frac{10mH}{10k} \quad = 1 \times 10^{-6} = 1.0\ \mu s$$

From the graph by using PSpice simulation, the time constant at 0.37 (37.0%) $V_{\max}$ is ;

$$\tau = 1.0 \mu s$$

The result can be represented in Fig.5 and we can defined that the time constant, $\tau = RC$ for this simulation is equal by using the calculation and PSpice simulation.

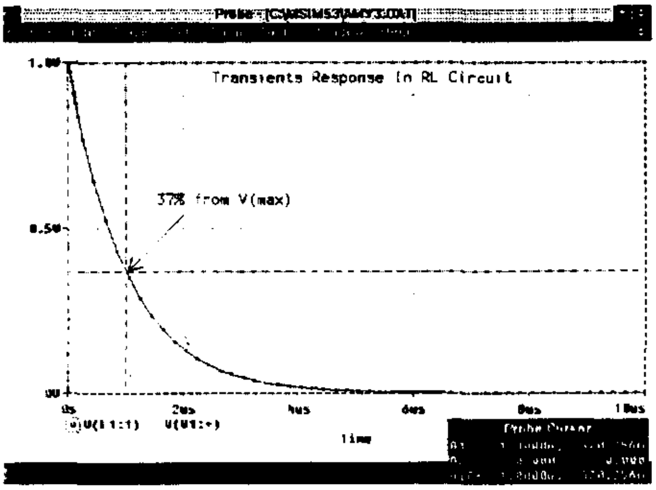


Fig.5 : Transients Response in RL Circuit

CHAPTER 4

4. TRANSIENTS RESPONSE IN RC CIRCUIT

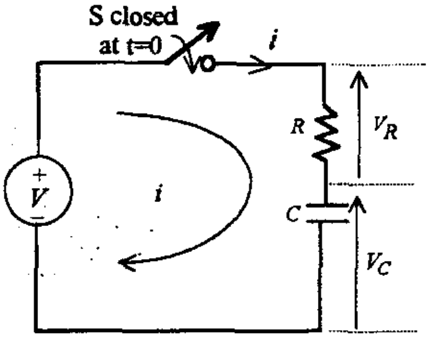


Fig. 6 : RC Circuit with arbitrary initial condition

The RC circuit comparises of resistance, *R*, capacitance, *C* and an excitation voltage V, as shown in Fig.6. The switch is closed at the t = 0, the current flows in the circuit, building a charge on the plates of the capacitor. Eventually, the current drops to zero as the capacitor voltage reaches the applied voltage, *E*.

4.1 Representation of the Circuit

The current in the circuit decreases in the form of a simple exponential, the rate of change of current is governed by the time constant, τ, of the circuit. Hence ;

$$i(t) = \frac{E}{R}\varepsilon^{-t/\tau} = \frac{E}{R}\varepsilon^{-t/RC} , \qquad (4.1)$$

where τ = time constant = RC (4.2)

The nature of the exponential is such that at the end of one time constant, that is, when , $t = \tau = RC$, the exponential $\varepsilon^{-t/\tau}$ has the value ;

$$\varepsilon^{-t/\tau} = \frac{1}{\varepsilon} = 0.37 \tag{4.3}$$

This causes the value of the current, $i(t)$ to be 37% of its maximum value of E/R and this can be represented as in Fig.7. It has dropped by 63%.

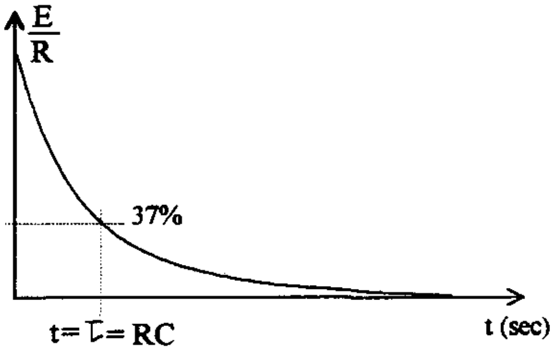


Fig. 7 : RC Circuit Response

The build-up of voltage across the capacitor begins at zero and reaches a maximum of E volts. The voltage $V_c(t)$ as a function of time is given by ;

$$V_c(t) = E[1 - \varepsilon^{-t/RC}] \tag{4.4}$$

Let us digress for a moment to observe the nature of the exponential function, $\varepsilon^{-t/\tau}$.

$$\text{At } t = 0,\ \varepsilon^{-t/\tau} = \varepsilon^{0} = 1 \tag{4.5}$$

$$\text{At } t = \infty\ ,\ \varepsilon^{-t/\tau} = \varepsilon^{-\infty} = \frac{1}{\varepsilon^{\infty}} = 0 \tag{4.6}$$

Thus, the value if the exponential can range between the limits of 0 and 1 as *t* goes from ∞ to 0 and represented as Fig. 8.

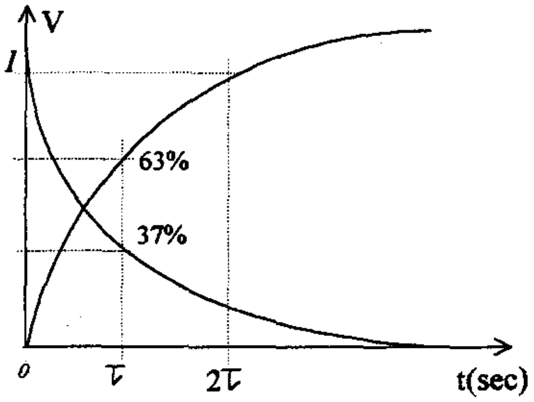


Fig.8 : Shows a graph of functions C

When t takes on the value of $t = \tau$, the exponential becomes ;

$$\varepsilon^{-t/\tau} = \frac{1}{\varepsilon} = 0.37 \tag{4.7}$$

Also, from the RC circuit, we can determine the time constant that occurs in this circuit by using calculation and simulation using PSpice.

4.2 PSPICE Simulation

In this circuit, we used R = 10 kΩ and C = 10 uF. Thus,

$$\tau = RC = 10\text{k} \times 10\mu F \;= 0.1 \text{ sec}$$

From the graph shown in Fig.9, by using PSpice simulation, the time constant at $V_{max} = 0.63$ (63.0%) is ;

$$\tau = 99.083 \text{ ms} = 0.099 \text{ sec} = 0.1 \text{ sec}$$

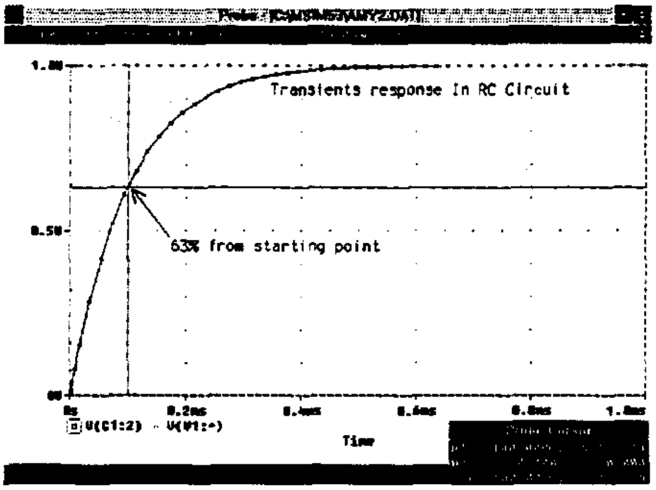


Fig.9 : Transients Response in RC Circuit

The result for this simulation represented as in Fig.9 and we can defined that the time constant is equal by using the calculation and PSpice simulation.

CHAPTER 5

5. TRANSIENTS RESPONSE IN RLC CIRCUIT

The simple RLC circuit is the basic model that is going to be used to represent a complex system.The resonant condition of the simpleR, L and C elements having a frequency response characteristic as shown in Fig. 10 below.

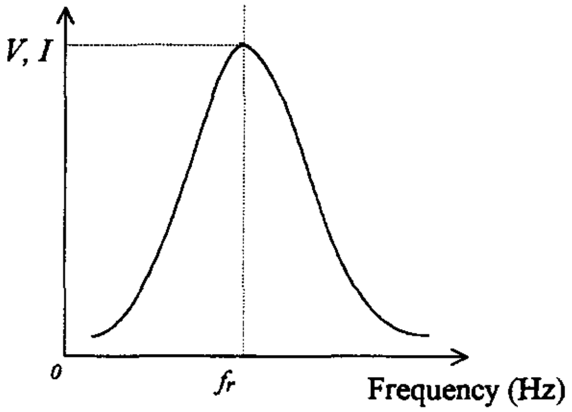


Fig. 10 : Resonance Curve

The resonant circuit selects a range of frequencies for which the response will be near or equal to the maximum. The frequency at which this occurs is called the *natural frequency*, f_r of the system.

When resonance occurs due to the application of the proper frequency (f_r), the energy absorbed by one reactive element is the same as that released by another reactive element within the system.

A resonant circuit must have a resistance, inductance and capacitance element as shown in Fig. 11 as below. The resistance element will always be present due to the resistance of the source, the internal resistance of the inductor , or an added resistance to control the shape of the response curve. The basic configuration for the serial resonant circuit appears in Fig. 11 with the resistive element listed as bellows ;

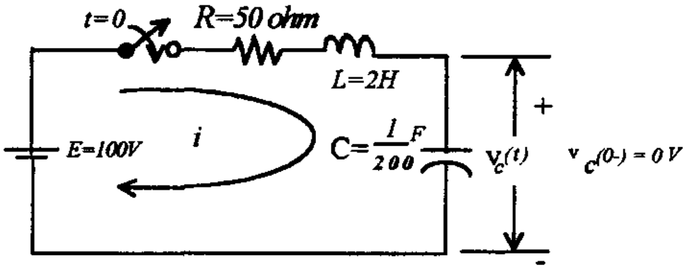


Fig.11 : Example RLC Circuit

The total impedance of this network at any frequency is determined by ;

$$Z_T = R + jX_L - jX_C$$

$$= R + j(X_L + X_C) \qquad (5.1)$$

The resonance condition occured when $X_L = X_C$. The total impedance at resonance is then simply $Z_{T_s} = R$ representing the minimum values of Z_T at any frequency. At resonance :

$$X_L = X_C \qquad (5.2)$$

$$\text{i.e } \omega L = \frac{1}{\omega C}$$

$$\omega^2 = \frac{1}{LC}$$

$$\omega = \frac{1}{\sqrt{LC}} \qquad (5.3)$$

$$\text{and } f = \frac{\omega}{2\pi} \qquad (5.4)$$

5.1 PSpice Simulation of a simple RLC circuit

The model consists of R= 10kΩ, L= 10mH and C= 10nF and as shown in Fig.12. The switch is closed at t = 0 and the voltage across C ic shown as in Fig. 13.

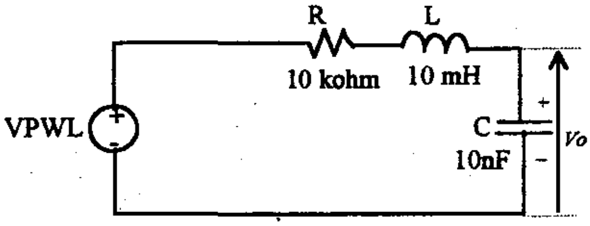


Fig. 12 : Basic Design Of the Project

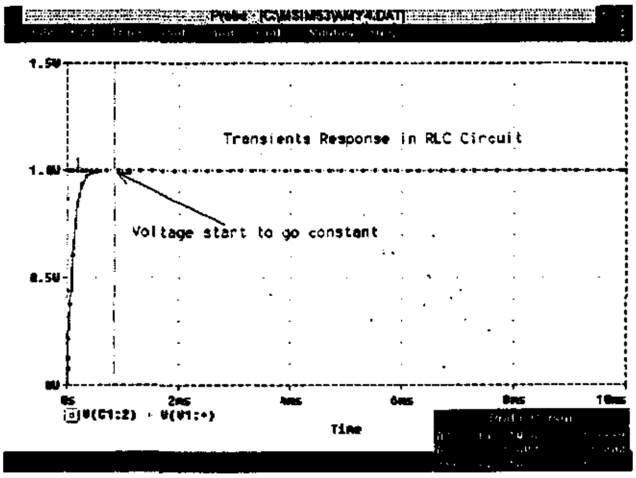


Fig. 13 : Transient Response in RLC Circuit

From the simulation, the voltage surge start from zero and rise up to the maximum value of V (≈ 0.999*V*) at t = 1 ms. After this the voltage continued to go constant . It is because due to the Piecewise linear specification.

CHAPTER 6

6. SWITCHING OF AN UNLOADED TRANSFORMER

The model of the system consists of :

- An ideal line, T with time delay TD and impedance Z_0 Ω. This represents the transmission line in the system
- An inductance with effective value L(Henry), which represent the unloaded transformer at the end of the transmission line.
- A capacitor, *C* which represent the effective capacitance to ground of the stub cable.

The typical system is as shown in Fig.14 below.

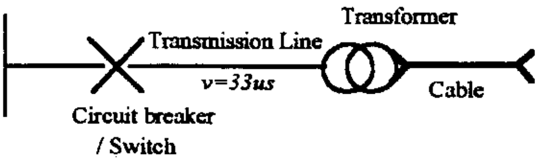


Fig. 14 : Typical simplified power system

6.1 Basic Model

The model respectively the system in Figure 14 is shown as in Figure 15.

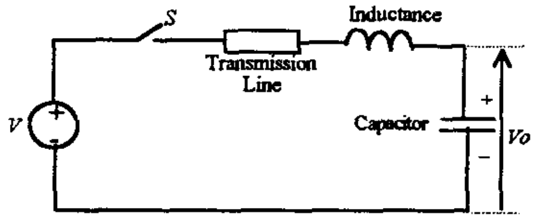


Fig. 15 : Basic Model of the Project

A typical simplified power system normally consists of a transmission line which is connected to a transformer at the end of the line. The line is protected by circuit breakers and the circuit breaker will operate when there is a fault in the line. During the opening the circuit breaker there is a tendency of high transient voltage generated at the transformer terminal if the transformer is unloaded. The simulation sets out to predict the resonance behaviour during switching of the unloaded transformer.

The voltage source, V represents the transient voltage that is generated by the switching on of the circuit breaker in the transmission line. When the surge travels the ideal line with Z_o and TD, the effects on the surge are; magnitude attenuation and time delay.

CHAPTER 7

7. PSPICE SIMULATION

This simulation are carried out to determine the combined an inductor, L= 10mH and a capacitor, C= 10nF the behaviour of transient voltage that appears where is connected in series. The voltage source is 1V DC Piecewise Linear at the end of an open circuit line.

7.1 Simulation 1 - Open circuit transmission line

The model of this simulation as shown in Fig.16.0 below.

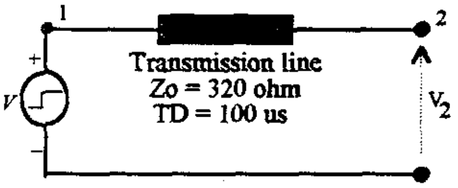


Fig. 16.0 : Open circuit transmission line

The transmission line with travelling time, TD= 100 us (length = 3000m) and the surge impedance, Zo= 320ohm. The voltage source is DC 1p.u. The characteristic of the open circuit tramnsmission line is achieved by connecting at the end of the section with a high resistor (1MΩ). The result of the simulation is shown in Fig. 16.1.

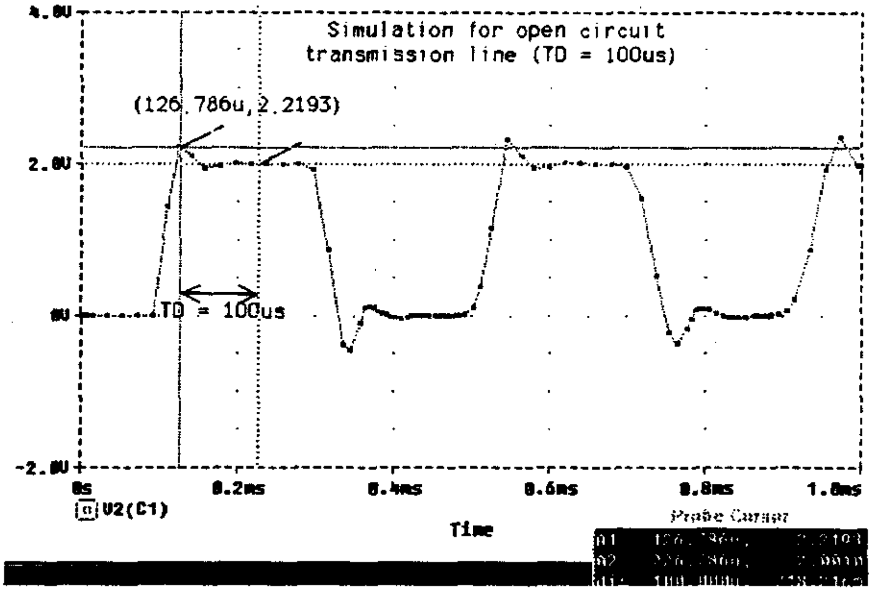


Fig. 16.1 : Simulation for open circuit transmission line

From this simulation the following conclusion can be deduced :

- The surge output voltage at node 2 with magnitude of reach to 2.2193 p.u is observed.

- The open circuit termination of the line gives the following voltage coefficient. Reflective koefficient $k_r = -1$ and transmission coeffiocient, $k_T = 0$.

- From the waveform, we can obtain the frequency at node 2 where $f_2 = \frac{1}{T_2} = \frac{1}{0.4ms} = 2500Hz$. Also, we can obtain the time delay, TD where $TD = \frac{1}{4 \times f_2} = \frac{1}{4 \times 2500Hz} = 1 \times 10^{-4}s = 100us$. This is satisfied for the simulation where used the time delay,TD is 100 μs.

- Thus, when the surge arrivaed at node 2, it is reflected back into the line making the magnitude of the surge two times the incident surge. Therefore we observed that the voltage in Fig. 16.2 is 2 p.u.

- TD of the incident surge is 100us. Therefore it takes 100μs to reach the end of the line. This effectively ' locked ' the voltage at the end of the line at 2 p.u for 200μs, until the next pulse reach the end of the line. Thus, when we have frequency, f_o at node 2 is reciprocal of 2 X 2TD.

$$\text{i.e}: f_o = \frac{1}{2X2TD} = \frac{1}{4TD}$$

When reduce the value of TD= 50us, the waveform for the simulation can be represented is shown in Fig. 16.2.

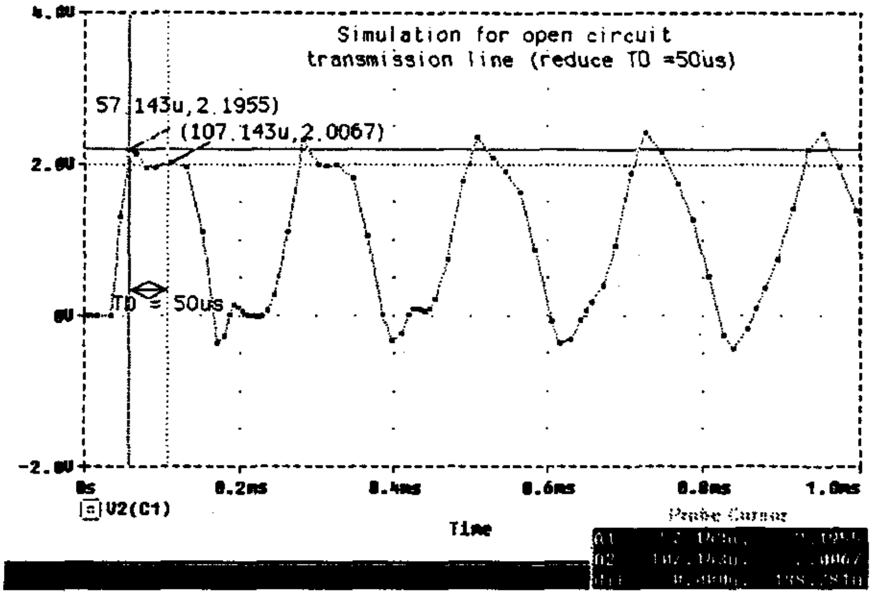


Fig 16.2 : Simulation for open circuit transmission line when reduce the value of TD= 50us

From this simulation the following conclusion can be deduced :

- When the value of TD is reduce, the frequency of the waveform is increases. For example the frequency of this simulation, $f_2 = \frac{1}{0.2ms} = 5000Hz$. Also, from this simulation we can obtain the value of time delay where $TD = \frac{1}{4 \times f_2} = \frac{1}{4 \times 5000Hz} = 5 \times 10^{-5}s = 50us$. This value is satisfied for the simulation where used TD is $50\mu s$. So, we can conclude that the waveform of the open circuit transmission line is respected to the time delay(TD) for transmission line.

- Also at node 2, the surge voltage reach to 2.1955 pu in 0.05ms and the transmitted voltage k_T = 0 and the reflected voltage k_r =-1, thus the effective voltage at the t= 0.05ms is the sum of the incident voltage that occurs in this section.

When increase the value of TD = 300us, the waveform for the simulation is represented as in Fig. 16.3.

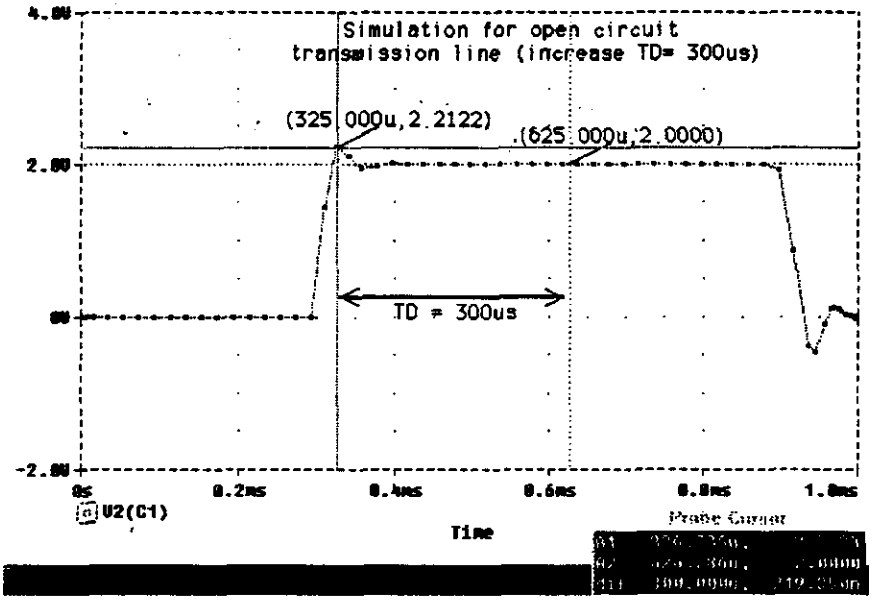


Fig 16.3 : Simulation for open circuit transmission line when increase the value of TD = 300us

From this simulation the following conclusion can be deduced ;

- When the value of TD is increase, the frequency of the waveform is reduce. For example the frequency of the simulation, $f_2 = \frac{1}{1.2ms} = 833.33Hz$. Also, we can obtain the value of time delay, $TD = \frac{1}{4 \times f_2} = \frac{1}{4 \times 833.33Hz} = 300us$ where this value is satisfied for this simulation where used TD= 300us.

- Also at node 2, the surge voltage reach up to 2.0067 pu in 0.3ms and the transmitted voltage $k_T = 0$ and the reflected voltage k_r =-1, thus the effective voltage at the t= 0.3ms is the sum of the incident voltage occurs in this section.

- Also from the simulation, we can conclude that the waveform for each simulation for open circuit transmission line is respected to the time delay, TD to give the different values of frequency that occurs at open circuit cable.

7.2 Simulation 2 - Resonance condition of an unloaded transformer

The objective of this simulation is to measured the maximum voltage at the stub cable/unloaded cable where L = 10mH, C = 10nF and TD is varied. The model is shown as Fig.17.0 below ;

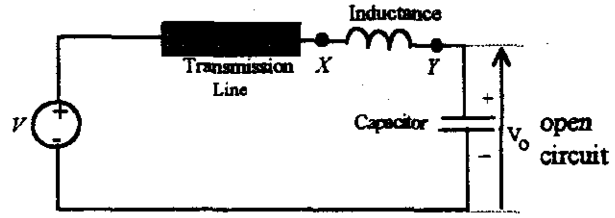


Fig.17.0 : Simulation for determine the voltage maximum at the open circuit cable

An example of this simulation is shown Fig.17.1,17.2 as below. This are the example simulation using the PSPICE software.

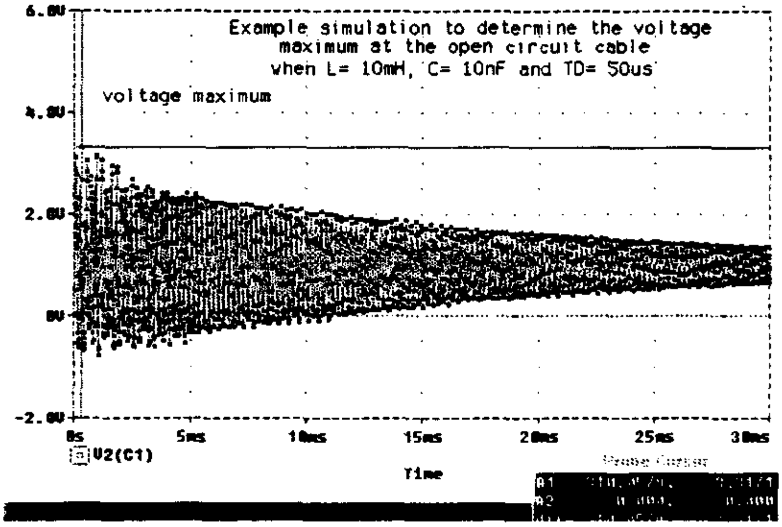


Fig.17.1 : Example simulation to determine the voltage maximum at the open circuit cable when L=10mH, C= 10nF and TD= 50us

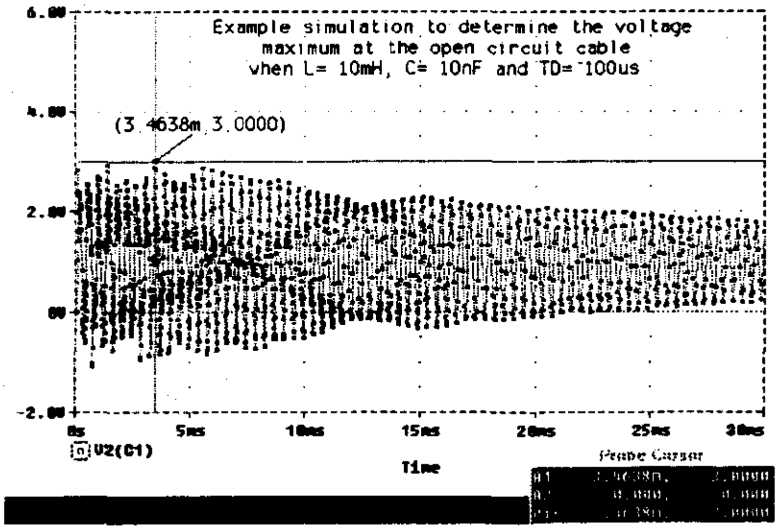
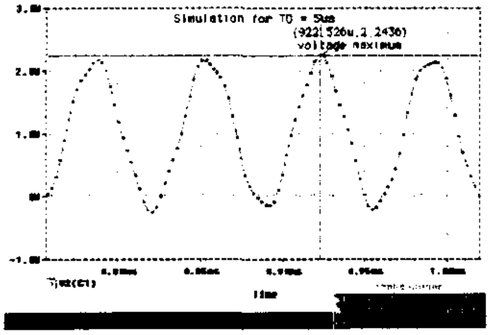


Fig.17.2 : Example simulation to determine the voltage maximum at the open circuit cable when L= 10mH, C= 10nF and TD= 100us

The simulation were repeated with different values of TD and all the equivalent value of the voltage maximum across the capacitor will be measured. The results are shown as in Fig. 17.3, 17.4, 17.5 and 17.6.



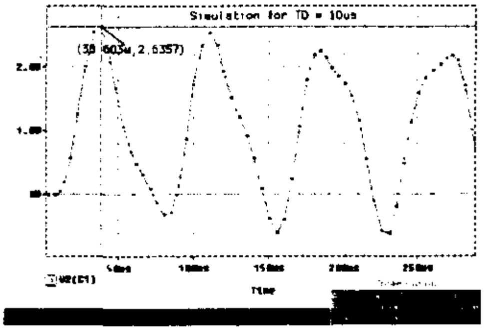


Fig.17.3 : Simulation for TD = 5us

Fig.17.4 : Simulation for TD = 10 us

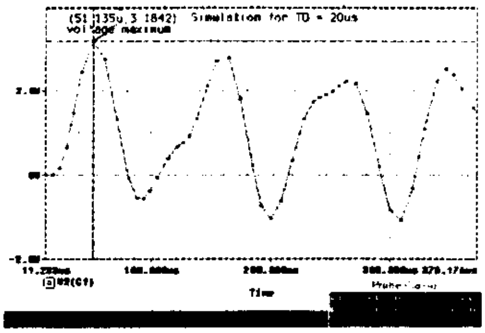


Fig. 17.5 : Simulation for TD= 20us

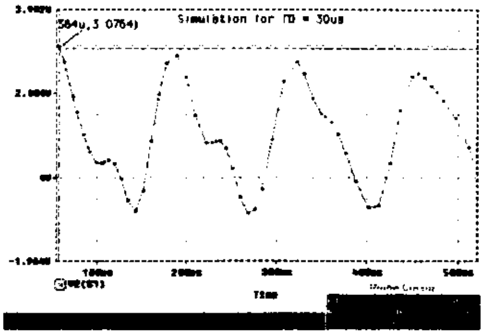
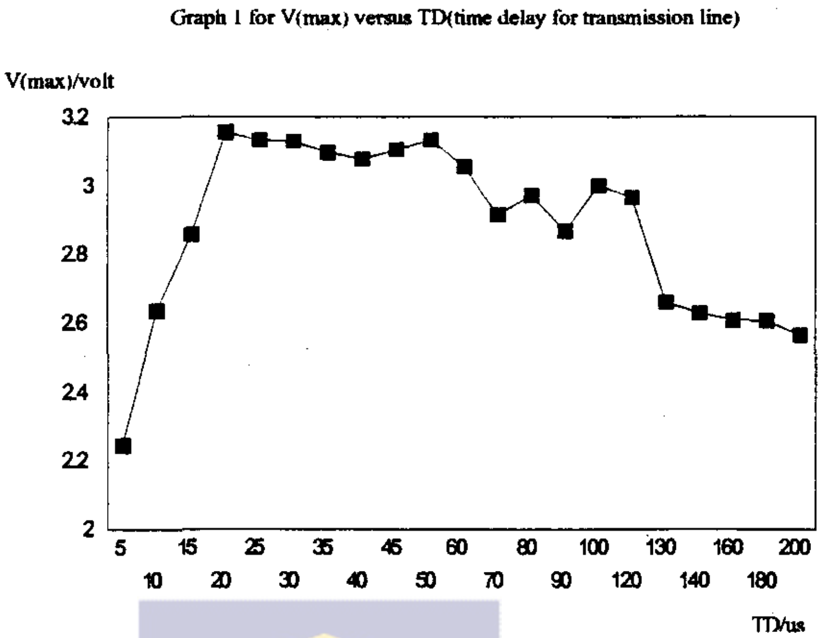


Fig 17.6 : Simulation for TD = 30us

The graph for V(maximum) across TD is represented as graph 1. Also the value of the simulation is represented in table 1.

TD (us)	V(max) p.u
5	2.24
10	2.64
15	2.86
20	3.15
25	3.13
30	3.13
35	3.1
40	3.1
45	3.1
50	3.13
60	3.05
70	2.91
80	2.97
90	2.87
100	2.99
120	2.96
130	2.66
140	2.63
160	2.61
180	2.61
200	2.58

Table 1: The voltage magnitude at the open circuit transformer terminal when TD is varied



Graph 1: Voltage maximum,V(max) versus time delay,TD

From this simulation the following conclusion can be deduced :

- Ideally, when TD is varied and frequency is varied so that the elements of the R,L and C were matched, the graph was obtained as the resonance responce characteristic.

7.3 Simulation 3 - Simulate the Model of the System

Simulation 3 is simulation for the model of the system . Also this simulation is open circuit cable. The objective of this simulation is to investigate the conditions of the transient voltage that occurs at the unloaded transformer terminal and also to establish the effect of different line termination on the surge voltage. This investigate is the combination of inductor and capacitor in series. The model of this simulation as shown in Fig.18.0.

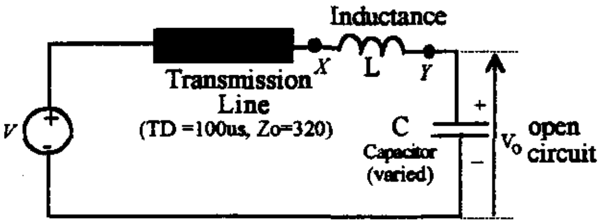


Fig.18.0 : Simulation the model of the system

Surge impedance and velocity of travelling wave of ideal transmission line can give the ratio of V/I = Z_0 = $\sqrt{L/C}$ where this is called surge impedance. Also this is known as the surge velocity, U. In ideal transmission line Z and U are dependent to the inductance and the capacitance.

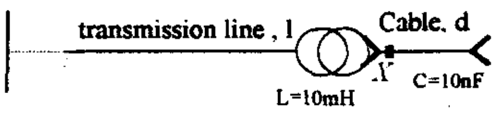


Fig. 18.1 : Representing voltage at X

From the Fig. 18.1, we can determine the value of d since we known the value of U. For example if the value of TD = 30us and since U = 300 m/us ;

From the formula :

distance, d = speed x time

So, d = t x U

= 30us x 300 m/us

= 9000m = 9 km

This simulation is also to simulate the RLC circuit. This simulation is referred at resonance condition. The line frequency, can be obtained at node X by varying TD of open circuit line. This simulation will attempt to match the frequency at node X by varying TD of the line to the resonance frequency of the RLC circuit, when both of this is equal the voltage across C (node Y) is at maximum value. Referred from the transformer with stub cable at resonance, all the values of capacitance and inductance are referred to 132 kV side of the transformer. The example simulation of this section were represented in Fig.18.2 and 18.3 below.

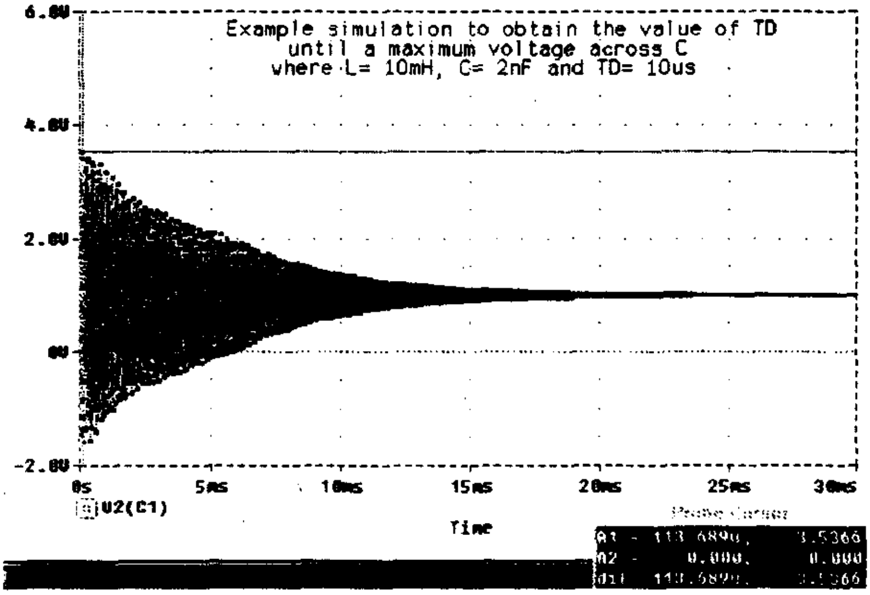


Fig.18.2 : Example simulation to obtain the value of TD until the a maximum voltage across C where L= 10mH, C= 2nF and TD= 10us

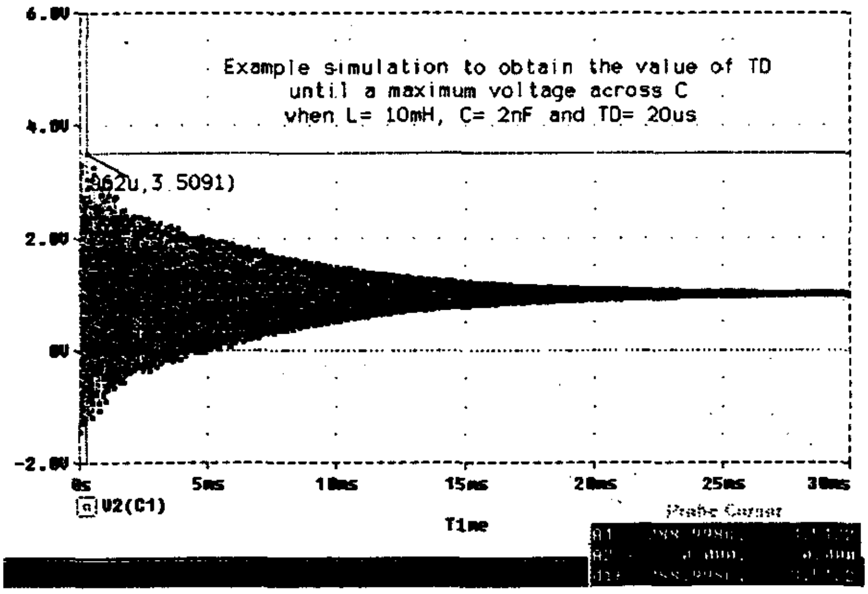


Fig. 18.3 : Example simulation to obtain the value of TD until a maximum voltage across C when L= 10mH, C= 2nF and TD= 20us

For each value of C, TD is varied to different d between circuit breaker and the transformer until a maximum voltage across C. Referred to the Fig.18.4, 18.5, 18.6 and 18.7, TD can obtained at the maximum value at C. As example voltage at C are increase from 2.5488V(TD= 5 us) to maximum value of 3.5287V(TD= 10 us) and then decreases as TD if increased C. So, take the TD= 10us.

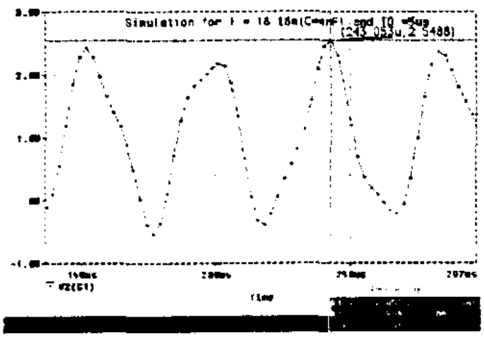


Fig. 18.4 : Simulation for TD = 5us

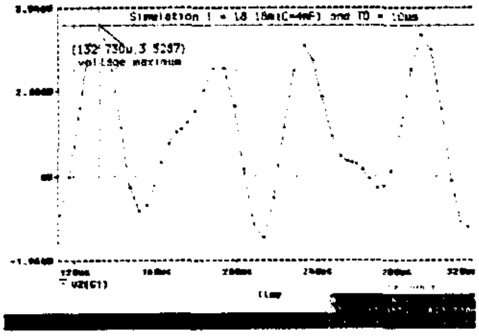


Fig. 18.5 : Simulation fro TD = 10us

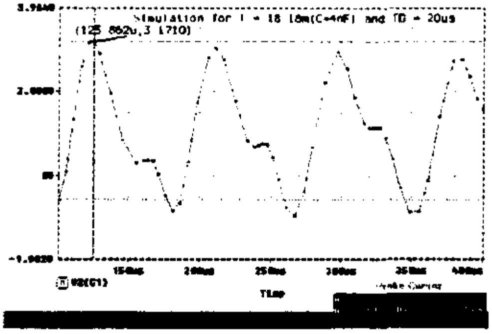


Fig.18.6 : Simulation for TD = 20us

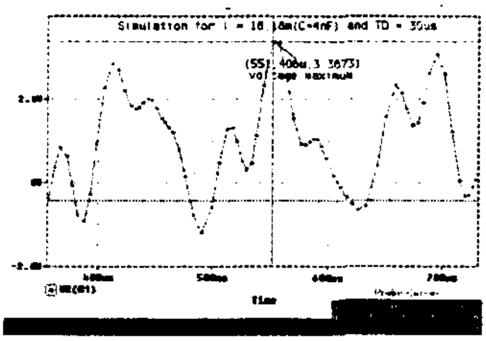


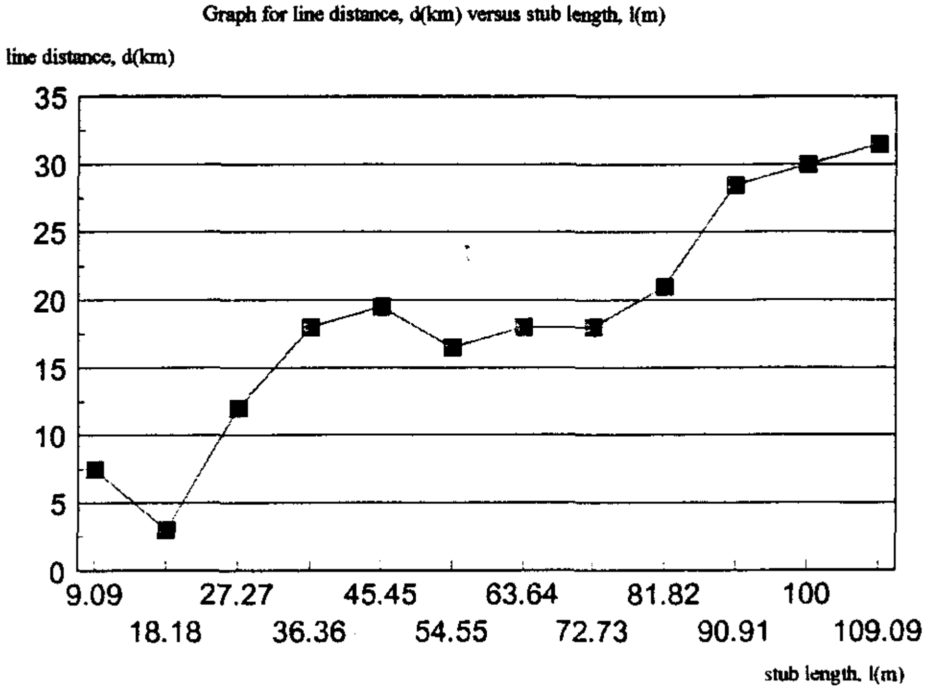
Fig.18.7 : Simulation for TD = 30us

Also we can calculate the value of *l* for stub length if we know the value of capacitor. For example, let say the value of capacitor is, C = 10 nF. So we can determine *l* as respectively from Fig18.1. So, from this we can determine the line distance, *d* of transmission line when varied value of TD.

Although, we can measured the value of cable length, *l*. The procedure are repeated with the different value of C and all the equivalent value of this simulation can be represented in the table 2. By using this value we can plot the graph of the relationship between cable length, *l* and the line distance, *d*. This can be represented as in table 2 and graph 2.

Stub length l, (m)	*Stub capacitor effective to 132 kV (nF)*	*LC resonance frequency, fo (kHz)*	*Cable Delay From PSPICE TD, (us)*	*line frequency f1, (kHz)*	*line distance (km)*
9.09	*2.0*	*35.59*	*25*	*10.0*	*7.5*
18.18	*4.0*	*25.16*	*10*	*25.0*	*3.0*
27.27	*6.0*	*20.54*	*40*	*6.25*	*12.0*
36.36	*8.0*	*17.79*	*60*	*4.17*	*18.0*
45.45	*10.0*	*15.91*	*65*	*3.85*	*19.5*
54.55	*12.0*	*14.53*	*55*	*4.55*	*16.5*
63.64	*14.0*	*13.45*	*60*	*4.17*	*18.0*
72.73	*16.0*	*12.58*	*60*	*4.17*	*18.0*
81.82	*18.0*	*11.86*	*70*	*3.57*	*21.0*
90.91	*20.0*	*11.25*	*95*	*2.63*	*28.5*
100.0	*22.0*	*10.73*	*100*	*2.50*	*30.0*
109.09	*24.0*	*10.27*	*105*	*2.38*	*31.5*

Table 2 : Simulation 3 for line distance/ d(m), stub length/l(m), fo, f1 and TD



Graph 2 : Line distance, *d*(km) versus stub length, *l*(m)

From this simulation the following conclusion can be deduced :

- Also the table and the graph was represented and we can conclude that the magnitude of the transient switching voltage is depended on the line distance, *d*. Also, we can state that the peak voltage at the cable and the ground is reduce as the impedance Zo(320) is increase.

CHAPTER 8

9. CONCLUSION

Switching of distribution cables by loadbreak(unloaded cable) and switching-compartment disconnects result in repetitive arc restrikes and prestrikes. Each arc ignition initiates voltage transients characteristic by high frequencies in the tens of hundreds of kHz and steep wavefronts with risetimes less than 1 μs.

Surge impedance discontinuities on the surge side of the switching location play a significant role in determining switching surge overvoltage magnitudes. Transient damping has been found to be primarily the result of nonlinear arc resistance.

Restriking does not lead to voltage escalation when load-break elbows and disconnects are operated because the resulting arc does not extinguish at transient current zeros, but continues until the transient oscillations are damped. This traps a voltage on the switched cable which is essentially the same as the instantaneous source voltage.

From the simulation, all the results were already discussed at the end of each simulation and the followings are the summary of the main points :

☐ Simulation for RL and RC circuit, which the elements contents are store energy, the capacitor in electrical field and the inductance in its magnetic field. Since energy is related to voltage and current, this unit of on time constants describes the manner in which the voltage and current reach a specific value. Furthermore, this specific time depends on the values of the resistance and capacitance or inductance. From this simulation we can measured that the time constant are equal by using calculation and PSPICE simulation.

☐ For Simulation 1, ideally the output of this simulation is the square wave where the surge voltage occurs during the operation. This is because due to the PWLS specification. Also we can determine the time delay, TD since we know the frequency where the time delay is matched due to the waveform.

☐ For Simulation 2, the value of the voltage maximum is not respect to the value of TD varied. It is because from the transients condition, particularly starting transient, can cause higher flux densities than exist in the steady-state condition especially at the small value of time. Also the graph was obtain is respect to the resonance responce characteristic.

☐ For Simulation 3, also the table and the graph was represented and we can conclude that the magnitude of the transient switching voltage is depended on the line distance, d. Also, we can state that the peak voltage at the cable and the ground is reduce as the impedance Zo(320) is increase.

Transient is initiated whenever there is a sudden change of circuit conditions. This most frequently occurs when a switching operation take place.[iii] The overvoltages caused by switching can cause high overvoltages at transformer terminal. When an unloaded transformer is switched on a cable feeder with a critical length, overvoltage can appear at the secondary terminals of the transformer which reach several times the peak value of the primary voltage.

The critical length of the connected cables is determined by the travel time or the resonant frequency of the primary cable and the resonant frequency of the transformer and secondary cable.

REFERENCES

[1] R.A. Walling, R.D.Melchior, B.A. McDermott

"Measuremet Of Cable Switching Transient In Underground Distribution System. - IEEE Transactions on Power Delivery, Vol. 10, January 1995

[2] Gerardus Chr. Paap, Abraham A. Alkema, Lao van der Sluis

"Overvoltage In Power Transformer Caused By No - load Switching "

IEEE Transactions on Power Delivery, Vol. 10, January 1995

[3] Y.H. Fu and G.C. Damstra

"Switching Transients During Energizing Capacitive Load by a Vacuum Circuit Breaker"

IEEE Transactions on Electrical Insulation, Vol. 28 , August 1993

[4] Dev Paul, Sriniva L. Venugopalan

"Power Distribution Equipment Overvoltage Protection"

IEEE Transactions on Industry Application, Vol. 30, No.5, September 1994

[5] L.Salazar, G.Joos

"PSPICE Simulation Of Three-phase Inverters By Means Of Switching Functions"

IEEE Transactions on Power Delivery, 1990

[6]. Allan Greenwood

"Electrical transients in power system"

Second edition

[7] Muhammad H. Rashid

"Spice for Circuits And Electronics Using PSpice"

Prentice-Hall international Editions, 1990

[8] Paul W. Tuinenga

"Computer- Aided Circuit Analysis Using PSPICE"

[9] Walter L. Weeks

"Transmission And Distribution of Electrical Energy"

Chapter 3, Pg. 67 - 85, 1981

[10] Dr. A.T. Johns, G.Ratcliff, Prof. A . Wright

"High Voltage Direct Current Transmission"

IEE Power Engineering Series 6, 1983, Chapter 8, Pg. 183 - 204

[11] Robert Boylestad, Louis Nashelsky

"Electric Devices And Circuit Theory"

Prentice-Hall International Editions, Fourth Edition, 1987

APPENDIX A

PSPICE DESCRIPTIONS

i. Acknowledgement

SPICE was developed by the CAD (Computer - Aided Design) Group of the University of California, Berkely, and is the sole property of the Regents of the University of California. The author is grateful to the University of California for granting permission to use excerpts from its Electronic Research Laboratory memoranda.

ii. SPICE Background Information

SPICE is an acronym for Simulation Program with Integrated Circuit Emphasis. It was developed by the Integrated Circuits Group of the Electronics Research Laboratory and Department of Electrical Engineering and Computer Science at the University of California, Berkely, California. The person credited with originally developing SPICE is Dr. Laaawrence Nagel, whose PhD thesis describes the algorithms and numerical methods used in SPICE. SPICE has undergone many changes since it was first introduced, improvements were and are being made on a continuous basis are encountered in using it.

iii. Process For Using SPICE

The process for using SPICE to analyses an electronic circuit is detailed as below. Fig.A is a flowchart showing the four necessary steps by using SPICE. The steps are :

a). Start with schematic diagram of the circuit, and notate it for SPICE. *Notation consists of three steps:*

i). Give each component, or circuit element, a name. For example, a 10K ohm resistor could be named R, R7, RLOAD or RBASEQ1. Notice that the name of a resistor must start with the letter R, and can contain from 1 to 8 character. Each element must have a different name. Capacitors start with C, inductors begin with L, and etc.

ii). Assign one node (a point of connection between two or more circuit elements) the node number zero. This is the ground (or datum) node, and all other circuit node voltage will be expressed with respect to this node. While the datum node need not be the actual circuit ground node, it is sensible to assign it this way.

iii). Each additional node in the circuit is given a node number, which nodes are numbered is arbitrary, and node numbers need not be sequential. Each node must be connected to at least two elements, except for *transmission lines and* MOSFET *substrates.*

b). Decide what type(s) of analysis you want to perform on the circuit. SPICE can do DC, AC transient, DC transfer function, DC small-signal sensitivity, distortion, noise, and Fourier analyses. Based on the types of analyses to be done, one or more control lines will have to be added to the input file. A control line could specify the values of DC voltage or current for a source, the range of frequencies an AC source is to have, or the time interval over which a transient analysis is to take place and the time step size to be used. Control lines also instruct SPICE how to present the results of the analyses, tabular form and graphical form are available.

c). Create an input file for SPICE using a text editor. Use only capitals latter in this file, though some SPICE derivatives will accept lower-case letter, others will not. The first line of the input file MUST be a title line. The last line of the input file MUST be .END

d). Run SPICE, which will perform a circuit analysis and create an input file. Examine the output file. If the result is not satisfactory, revise the circuit as necessary, edit the input file to reflect the change, and run SPICE again.

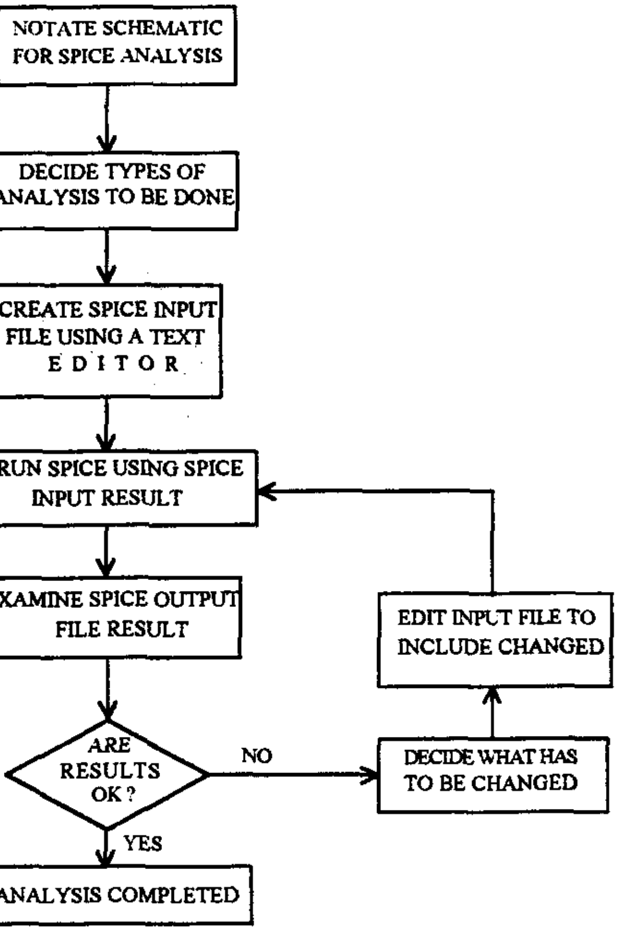


Fig.A : Flowchart Showing How to Analysisa Circuit with SPICE

iv. Passive Elements Using PSpice

PSpice recognize passive elements by their symbols and their models. In this project the elements can be switch, resistor (R), capacitor (C), inductor (L) and lossless transmission line. The models are necessary to take into account the parameter variations. For example, the value of a resistor depends on the operating temperature.

In this project the simulation of passive element on PSpice requires specifying the following:

a). Switch

PSpice allows simulation of a special kind of switch whose resistance varies continuously depending on the voltage or current. When the switch in on, the resistance is Ron, and when it is off, the resistence becomes Roff. Two types of switches are in PSpice are :

i). Voltage-controlled switch
ii). Current-controlled switch

b). Resistor

The symbol for the resistor is R. The name of a resistor must start with R and it take the general form of

R⟨name⟩ N+ N- RNAME VALUE

A resistor does not have a polarity and the order of the nodes does not matter. However, by defining N+ as the positive node and N- as the negative node, the current is assumed to flow from node N+ through the resistor to node

N-. RNAME is the model name that defines the parameters of the resistor. VALUE is the nominal value of the resistance.

c). Capacitor

The symbol for a capacitor is C. The name of capacitor must start with C and it takes the general form of

R⟨name⟩ N+ N- CNAME VALUE IC=VO

N+ is the positive node and N- is the negative node. The voltage of node N+ is assumed positive with respect to node N- and the current flows from node N+ through the capacitor to node N-. CNAME is the model name and VALUES is the nominal value of the capacitor. IC defines the initial (time-zero) voltage of the capacitor Vo.

d). Inductor

The symbol for an inductor is L. The name of an inductor must start with L and it takes the general form of

R⟨name⟩ N+ N- LNAME VALUE IC=IO

N+ is the positive node and N- is the negative node. The voltage of N+ is assumed positive with respect to node N- and current flows from node N+

through the inductor to node N-. LNAME is the model name and VALUE is the nominal value of the inductor. IC defines the initial (time-zero) current of the inductor, Io.

e). Lossless Transmission Line

The symbol for a lossless transmission line is T. A transmission line has two port - (input and output). The general form of a transmission line is ;

$$T\langle name\rangle \quad \mathrm{NA+} \quad \mathrm{NA-} \quad \mathrm{NB+} \quad \mathrm{NB-} \quad \mathrm{ZO}{=}\langle value\rangle \quad [TD = \langle value\rangle]$$

$$[F = \langle value\rangle \quad \mathrm{NL}{=}\langle value\rangle]$$

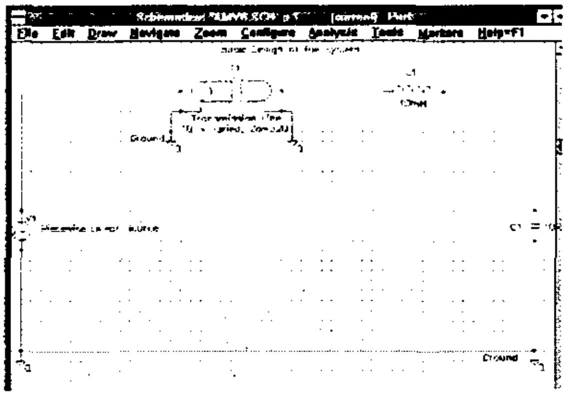
T$\langle name\rangle$ is the name of the transmission line. NA+ and NA- are the nodes at the input port. NB+ and NB- are the nodes at the output port. NA+ and NB+ are defined as the positive nodes. NA- and NB- are defind as the negative nodes. The positive current flows from NA+ to NA- and from NB+ to NB-. ZO is the characteristic impedance.

The length of the line can be expresssed in either of two form:

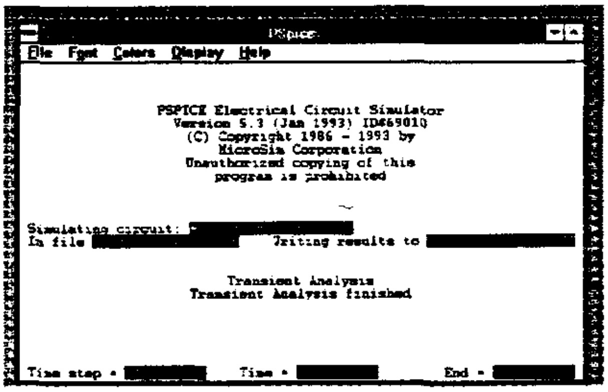
i). The transmission delay TD may be specified, or

ii). The frequency F may be specified together with NL, which is the normalized electrical length of the transmission line with respect to wavelength in the line at frequency, F.

APPENDIX B



Fig(a) : Draw the schematic diagram



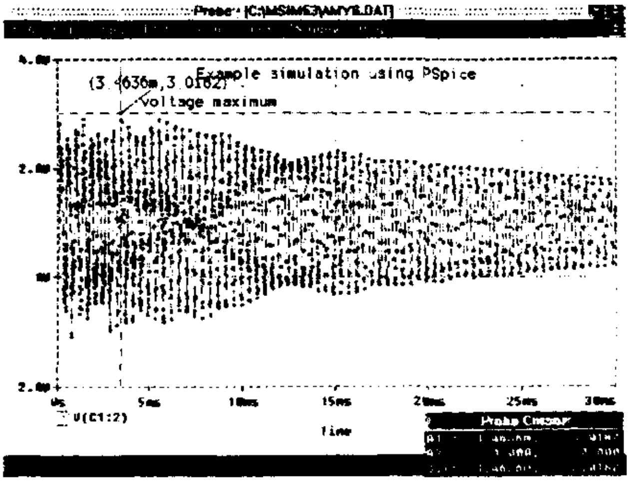
```
PSPICE Electrical Circuit Simulator
Version 5.3 (Jan 1993) ID469010
(C) Copyright 1986 - 1993 by
MicroSim Corporation
Unauthorized copying of this
program is prohibited

Simulating circuit: [redacted]
In file [redacted]          Writing results to [redacted]

Transient Analysis
Transient Analysis finished

Time step = [redacted]     Time = [redacted]          End = [redacted]
```

Fig(b) : Do the Simulation



Fig(c) : Simulation Result