IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

Impact Factor: 7.57

Volume 4, Issue 2, January 2024

A Study on the Effect of TVS Diode in a Transmission Line with RLC Series Load and PWL Signal Input

Anupendra Singh

Sr.Lecturer, Department of Electronics and Telecommunication,

Government Women Polytechnic College, Jabalpur, Madhya Pradesh, India

Abstract*: Transmission lines are of different types and different types of loads are connected across it as per application. Transmission lines respond differently for different types of loads connected across its terminals. Ideally we want original signal to be transmitted through transmission line and identical signal received at receiver end. Due to different reasons this signal gets distorted and original signal is not received at receiver terminal.*

Voltage fluctuations taking place due to various reasons affect the original signal and distort it. Fluctuations arise due to numerous reasons. Lightning and electric and magnetic effects of nearby transmission lines are a few common reasons for signal distortion. Effect of these voltage fluctuations is to be minimized. TVS diode is one such tool to minimize the effects of voltage fluctuations. In this paper we have simulated a transmission line with RLC series load. A TVS diode is connected across the RLC series load and an Piece Wise Linear(PWL) signal is applied across the transmission line input terminals and responses of the transmission line at different node points are studied. Simulation results at different node points of the circuit are recorded in JPEG image form and published in this paper.

Keywords: LT spice software , transmission line ,TVS diode, simulation

I. INTRODUCTION

TVS diode is transient voltage suppression diode which is widely used in electronics industry. TVS diode is normally used to avoid voltage fluctuations and transient voltage surges. This diode protects electronics circuit and components from sudden voltage fluctuations in the event of lightning , electrostatic discharge, ESD or inductive switching. These diodes clamp High Voltage spikes to safe level by diverting access voltage from the protected circuit . TVS diodes are of two types Uni directional and by directional. Normally unidirectional TVS diodes are used for DC circuits and by directional TVS diodes are used for AC circuits. Voltage level is decided as per application. TVS diodes have fast response time. TVS diodes are used for protection in USB ports , communication lines and ICS. Also used for lightening protection and protection of circuits used in Auto motive electronics. Parameters which define TVS diode include breakdown voltage ,clamping voltage, peak pulse power ,working voltage etc.

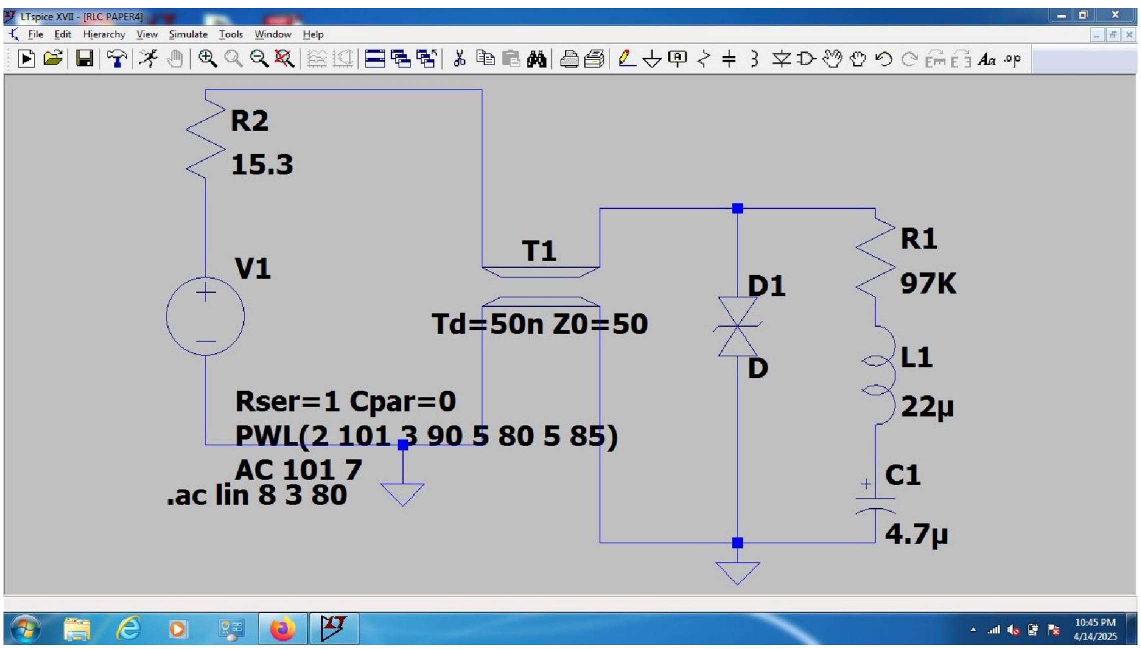
II. REVIEW OF LITERATURE

LT spice is a powerful simulation software used in electronics industry for simulating circuits and analyzing performance of analog devices .Simulation helps to predict performance of a circuit before its final hardware implementation. It is a free and user friendly software which allows engineers to simulate and test desired electronic circuit and models. Full form of Spice is simulation program with Integrated circuit emphasis. This simulation software is useful for simulation of both analog and digital circuits . It is used in range of electronic devices and circuits including transmission lines. LT spice has very good user interface and it has a graphical user interface GUI also which is very useful. This software is a high speed simulation software and useful for fast circuit simulation both for simple as well as large and complex circuit designs. It's interface is smooth making it very convenient for real time analysis with the help of real time waveform plotting and analysis of current, voltages and power in different electronics circuits and components. This software has very powerful model library of components including the range of electronic devices

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.57
Volume 4, Issue 2, January 2024

including transistors ,registers, inductors, diode, opamp and different types of voltage sources. There is a possibility for the user to customize components as per requirement. This software has advanced features also like parametric sweeps fourier analysis and noise analysis. This software is also very suitable for multi stage circuit designs and simulations in Power Electronics designs. Key elements of LT spice interface include schematic window . schematic window is place where the circuit is built and simulated. It represents complete circuit including its components and inter connections . This software has a component library from where different components as per the requirement could be draggged and brought into the schematic window to design the circuit . This software has a plot window where simulation is done and the results are plotted thus displaying the simulation results in the form of waveforms of voltage, current or any parameter which is of interest. This software has simulation command bar where type of simulation which is intended is decided for example transient analysis, DC operating point and AC analysis. It has toolbar which contains different functions like selecting components ,running simulation and plotting results. It has error log where log of errors occured during simulation and testing are recorded.

III. CIRCUIT DIAGRAM



. Fig1 .Circuit Diagram

Findings:

Result of this simulation and response at different node points of the circuit are recorded in JPEG image form and analyzed. Response at different node points of the circuit represent attenuation in DB ,frequency in Hertz and phase in degrees. Different node points chosen to study the response include one above the TVS diode ,across load resistance and load inductance etc. The recorded response in JPEG image form is reproduced below as findings of this simulation

IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

Impact Factor: 7.57

Volume 4, Issue 2, January 2024

1. NODE ABOVE TVS DIODE

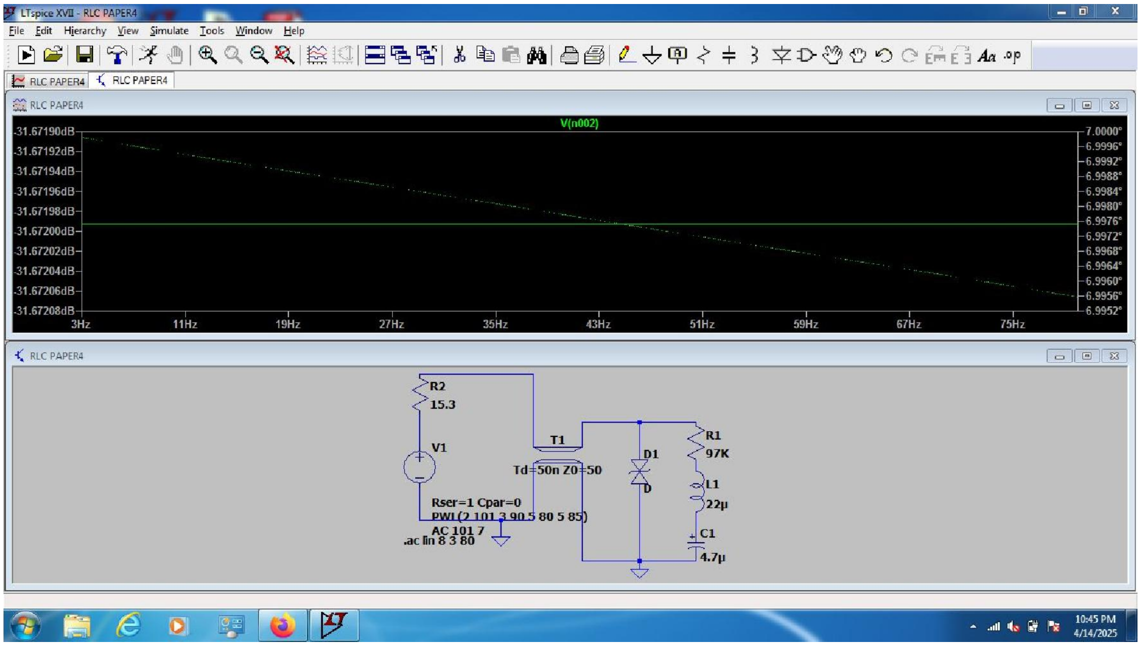


Fig 2. Simulation result at node above TVS DIODE

2. ACROSS CAPACITOR C1

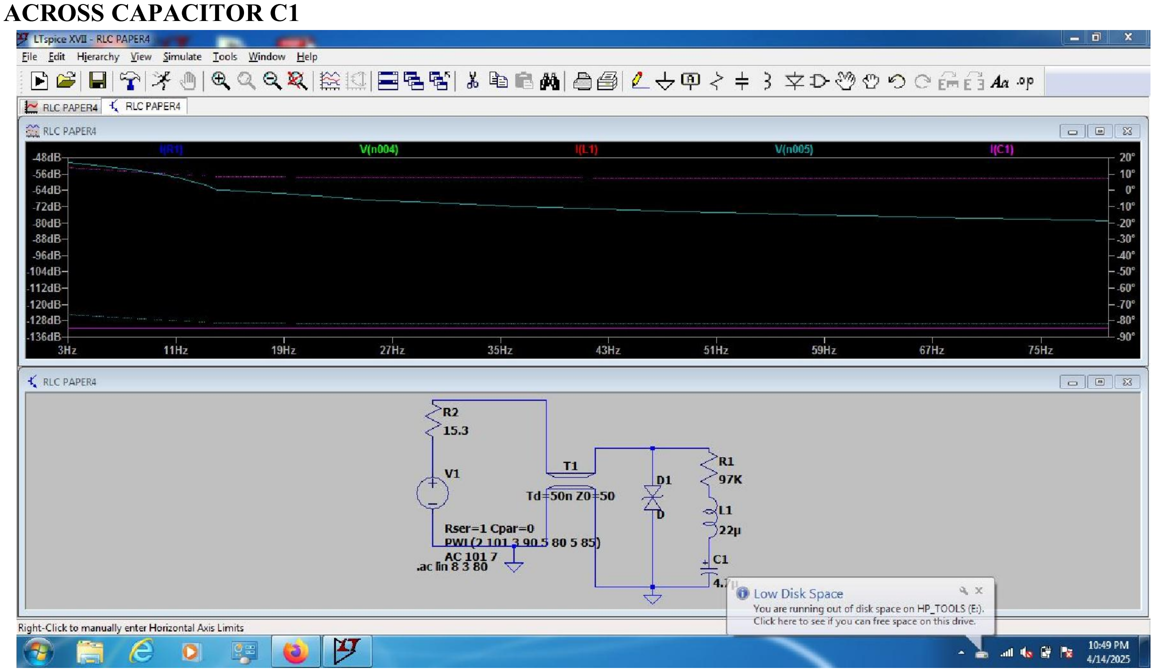


Fig3. Simulation result across capacitor C1

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.57
Volume 4, Issue 2, January 2024

3. ACROSS RESISTANCE R1

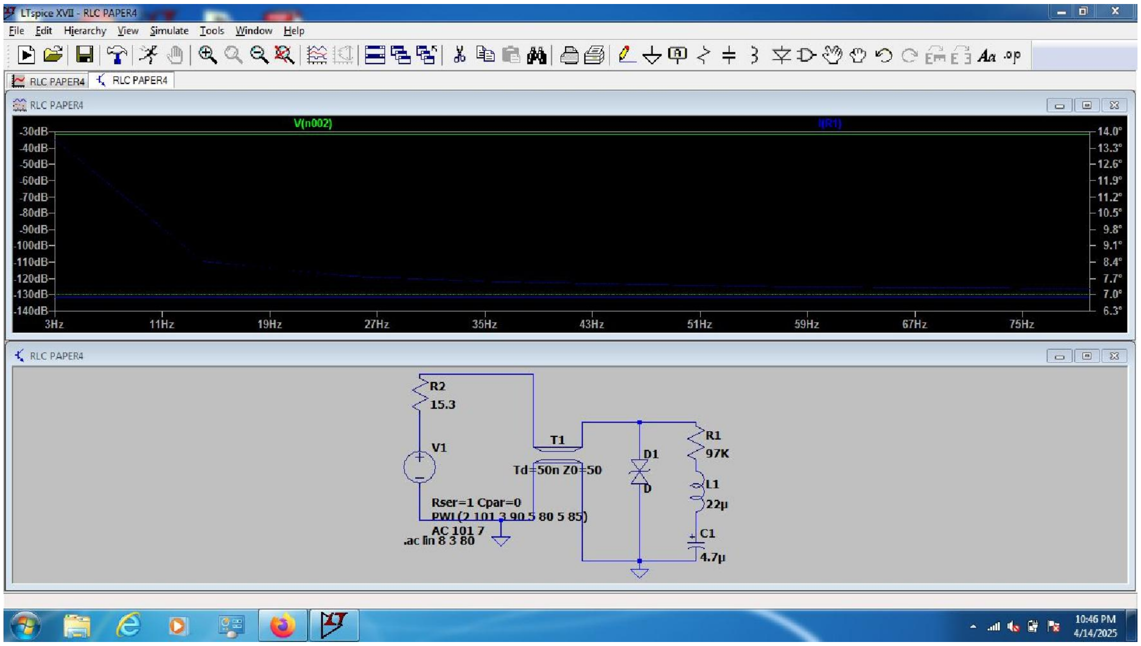


Fig 4. Simulation result across resistance R1.

4. BETWEEN L1 AND C1

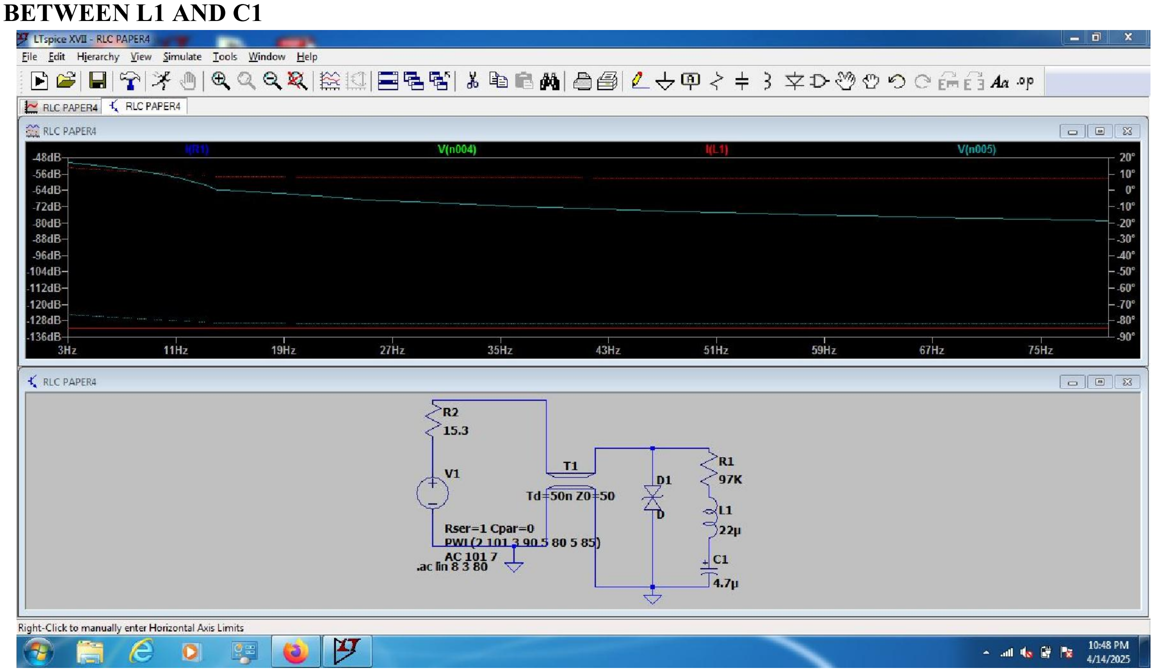


Fig 5. Simulation result between L1 and C1

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.57
Volume 4, Issue 2, January 2024

5. BETWEEN R1 AND L1

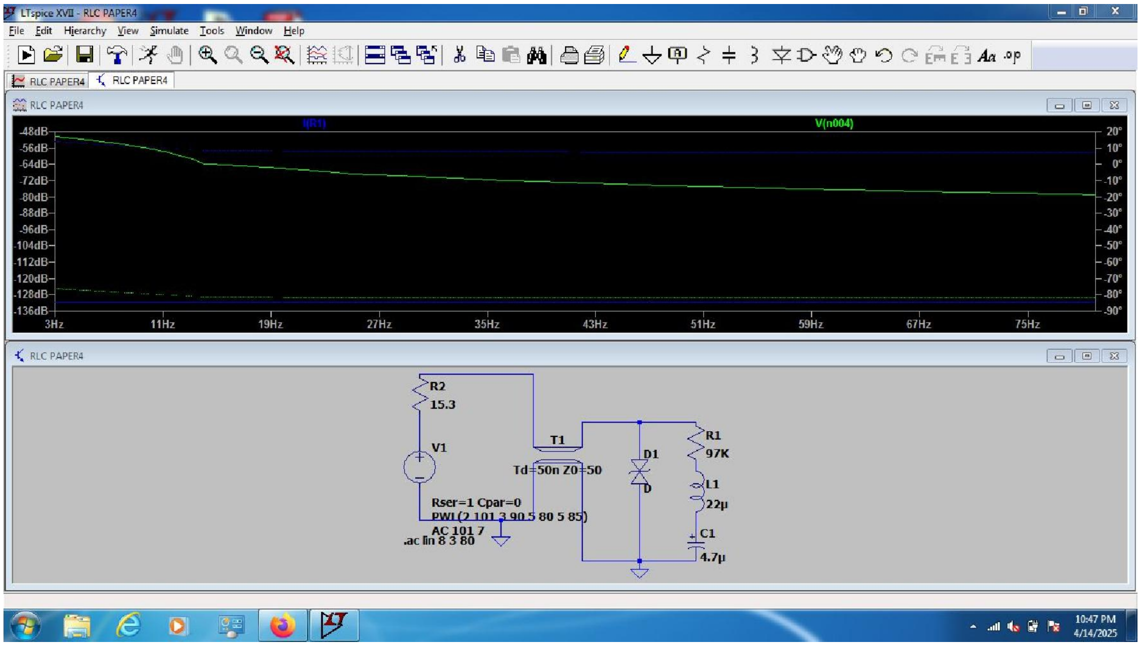


Fig 6. Simulation result between R1 and L1.

REFERENCES

[1]. White, K. Preston; Ingalls, Ricki G. (December 2015). "Introduction to simulation". 2015 Winter Simulation Conference (WSC). pp. 1741–1755.

[2]. Sokolowski, J.A.; Banks, C.M. (2009). Principles of Modeling and Simulation. John Wiley & Son. p. 6. ISBN 978-0-470-28943-3.

[3]. Drakos, Nikos; Hennecke, Marcus; Moore, Ross; Swan, Herb (November 22, 2013). "Transmission Line". Quite Universal Circuit Simulator (Qucs).

[4]. Qian, Chunqi; Brey, William W. (2009). "Impedance matching with an adjustable segmented transmission line". Journal of Magnetic Resonance. **199** (1): 104–110.

[5]. Rhea, Randall W. (1995). HF Filter Design and Computer Simulation. McGraw-Hill, Inc. pp. 86–89. ISBN 0-07-052055-0.

[6]. LT Spice learning mannuals