Exploring Circuit Modeling and Radiation Effects using SPICE

Murali S Bharadwaja, Dr. Jamuna S
VLSI Design and Embedded Systems, Dayananda Sagar College of Engineering, Bangalore, India
Email : muralisbharadwaja@gmail.com

***Abstract* - The research paper investigates the impact of radiation on circuit modeling, specifically focusing on inverter and MOSFET characteristics using SPICE. The study delves into the effects of Single Event Latch-up (SEL) and Single Event Upset (SEU) on these components. Through simulation and analysis, this paper aims to provide a comprehensive understanding of how radiation influences the behavior of circuits, shedding light on potential vulnerabilities and strategies for mitigation in radiation-prone environments.**

Keywords - Circuit Modeling, Radiation Effects, SPICE (Simulation Program with Integrated Circuit Emphasis), Single Event Effects (SEEs), Total Ionizing Dose (TID)

1. INTRODUCTION

In today's rapidly advancing technological landscape, electronic circuits are at the heart of countless innovations, from the smartphones we use daily to the complex systems that power spacecraft exploring the cosmos. These electronic circuits must not only function reliably under normal operating conditions but also withstand the challenges posed by the harsh environment of outer space, as well as the potential hazards of radiation on Earth.

Circuit modeling is a fundamental practice in the field of electrical and electronic engineering. It enables engineers to design, analyze, and optimize circuits to meet the specific requirements of various applications. One powerful tool that has revolutionized the way engineers approach circuit modeling is the Simulation Program with Integrated Circuit Emphasis, or SPICE.

SPICE, which stands for Simulation Program with Integrated Circuit Emphasis, is a versatile software tool that enables engineers and researchers to simulate, analyze, and understand the behavior of electronic circuits. Originally developed at the University of California, Berkeley in the 1970s, SPICE has since evolved into a standard tool used in both academia and industry for simulating electronic circuits. It is essential for designing and testing a wide range of devices, from simple amplifiers to complex integrated circuits and even radiation-hardened electronics for space missions.

The primary objective of this exploration is to delve into the world of circuit modeling and its relationship with radiation effects, all through the lens of SPICE. This paper will take through the intricate realm of circuit simulation, understanding its fundamentals, and exploring its practical applications. Furthermore, it discusses how SPICE can be employed to investigate the behavior of electronic circuits in radiation-prone environments, shedding light on the critical aspect of radiation effects.

The influence of radiation on electronic circuits is a topic of utmost importance, especially in the context of aerospace, nuclear, and medical industries. Whether it's cosmic radiation in space missions, ionizing radiation in medical equipment, or the radiation background on Earth, the impact on electronic components and circuits can be profound. Understanding and mitigating these radiation effects is crucial for ensuring the reliability and longevity of electronics.

Throughout this exploration, this paper will touch upon:

Fundamentals of Circuit Modeling: This paper introduces the basic concepts of circuit modeling, including passive and active components, circuit analysis techniques, and how SPICE is employed to simulate electronic circuits.

SPICE: A Versatile Tool: This paper delves into the capabilities of SPICE and its version, such as LTSpice demonstrating it can be used for circuit design and analysis.

Radiation Effects on Electronic Circuits: A comprehensive discussion on the various types of radiation and their impact on electronic circuits, including single-event effects (SEE), total ionizing dose (TID), and displacement damage dose (DDD).

Using SPICE for Radiation Analysis: This paper explores how SPICE can be used to simulate and analyze the effects of radiation on electronic circuits, enabling engineers to design radiation-hardened electronics for space exploration and other critical applications.

This exploration will provide a comprehensive understanding of the interplay between circuit modeling, radiation effects, and SPICE simulations. It will equip engineers, researchers, and enthusiasts with the knowledge and tools required to design robust and resilient electronic systems capable of withstanding the challenges posed by radiation, be it in space or closer to home. As we embark on this journey, we will unlock the potential for innovation and reliability in the ever- expanding world of electronic circuits.

A. Motivation

The motivation behind "Exploring Circuit Modeling and Radiation Effects with SPICE" is multifaceted:

Aerospace & Space Exploration: To create radiation- resistant electronics vital for reliable space missions and satellite communications.

Nuclear Safety: Ensuring electronic control systems' reliability in nuclear facilities to prevent radiation- induced failures.

Medical Device Advancements: Developing safe and accurate electronic systems for medical imaging and impedance state, effectively latching up. This can lead to severe consequences, including device failure and therapy damage if not managed properly.

Robust Electronics: Extending electronic systems' Total Ionizing Dose (TID): It is a critical radiation effect resilience in radiation-prone environments like aviation and particle physics experiments. that pertains to the cumulative ionizing radiation exposure over time on electronic devices, integrated Scientific Progress: Advancing knowledge in radiation effects on electronics for technological growth. Innovation and Development: Fostering innovation by combining circuit modeling, radiation effects analysis, and SPICE simulations.

Data Integrity Assurance: Maintaining data integrity incritical sectors like telecommunications and data storage. Future Missions' Challenges: Preparing resilient electronics for upcoming radiation-intensive space missions. This exploration is propelled by scientific curiosity, technological necessity, and the pursuit of innovation to enhance electronic technology's safety, reliability, and advancement across crucial applications.

B. Terminologies:

Circuit modeling: Circuit modeling is a fundamental practice in the field of electrical and electronic engineering. It involves creating mathematical representations of electronic circuits to analyze, simulate, and predict their behavior. These models enable engineers and researchers to understand how different components and devices within a circuit interact and respond to various input signals. Circuit modeling is a crucial step in the design, analysis, and optimization of electronic systems, and it plays a pivotal role in both theoretical and practical aspects of electrical engineering.

Radiation effects: Radiation effects refer to the physical and electrical changes that occur in materials and electronic components when exposed to various forms of radiation. These effects can have significant implications in a wide range of applications, including nuclear technology, space exploration, medical devices, and even in the design of electronics for terrestrial use.

A Single-Event Upset (SEU): It is a type of Single Event

Effect (SEE) that occurs when a high-energy particle, typically a neutron or proton, strikes an electronic device, such as a semiconductor memory cell or a digital flip-flop within an integrated circuit. This interaction can cause a temporary and unintended change in the device's state, leading to an alteration in stored data or the logic state of the circuit. SEUs are particularly relevant in applications where radiation exposure is a concern, such as aerospace, nuclear facilities, and high-altitude aviation.

Single event latch up: Single-Event Latch-Up (SEL) is a critical and potentially catastrophic Single Event Effect (SEE) that can occur in electronic devices, particularly integrated circuits, when exposed to ionizing radiation, such as high-energy particles or heavy ions. SEL results from a high-energy particle striking a specific region within the device, triggering a parasitic thyristor-like structure, which causes the device to enter a low-impedance state, effectively latching up. This can lead to severe consequences, including device failure anddamage if not managed properly.Total Ionizing Dose (TID): It is a critical radiation effectthat pertains to the cumulative ionizing radiation exposure over time on electronic devices, integratedcircuits, and materials. Unlike transient radiation effects, such as Single-Event Effects (SEE), TID results from the gradual accumulation of ionizing radiation and is often associated with prolonged exposure to ionizing radiation sources. TID can significantly impact the performance and reliability of electronic components and systems,and it is a key consideration in various applications,particularly in the aerospace and nuclear industries.

SPICE: SPICE, which stands for Simulation Program with Integrated Circuit Emphasis, is a widely used open- source or commercial software tool for simulating and analysing electronic circuits. Developed initially at the University of California, Berkeley, in the 1970s, SPICE has since evolved into an industry-standard simulation program used for a wide range of electronic circuit design, analysis, and optimization tasks.

C. Tool used:

LTSPICE® is a powerful, fast, and free SPICE simulator software, schematic capture and waveform viewer with enhancements and models for improving the simulation of analog circuits. Its graphical schematic capture interface allows you to probe schematics and produce simulation results, which can be explored further through the built-in waveform viewer.

Fig1. LTspice logo

- asc - schematic. It consists of a netlist based on SPICE text- based commands.
- asy - electronic symbol shown in a schematic.
- cir - external netlist input.
- fft - FFT binary output.
- lib - model library subcircuits.
- plt - waveform viewer plot settings.
- raw - binary output, optional ASCII output.
- sub - subcircuit.
- lib / .sub / .mod / .model - device model. While any file extension is allowed, users tend to gravitate towards common ones.

2. METHODOLOGY

- Select the Circuit
- Collect Circuit Information
- Radiation Model Selection
- Modify Circuit for Radiation Effects
- SPICE Simulation Setup
- Radiation Scenarios
- Perform Simulations
- Analyze Results

3. CIRCUITS UNDER CONSIDERATION

A. RTL INVERTER

VCC 4 0 5

VIN 1 0 PULSE 0 5 2NS 2NS 2NS 30NS
RB 1 2 10K

Q1 3 2 0 Q1

RC 3 4 1K

- MODEL Q1 NPN BF 20 RB 100 TF .1NS
- CJC 2PF
- DC VIN 0 5 0.1
- TRAN 1NS 100NS
- END

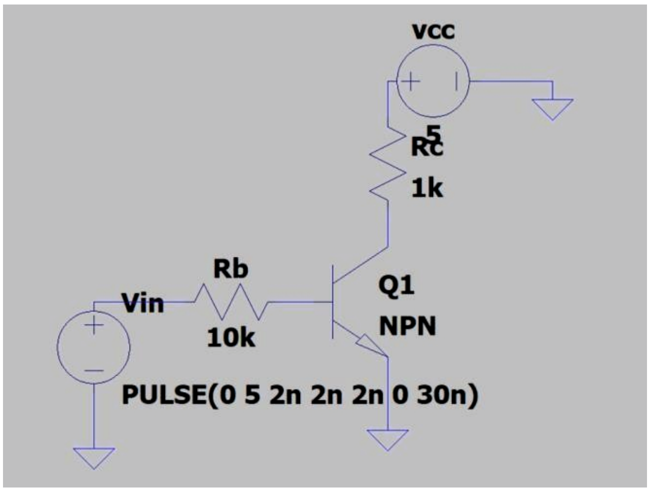

Fig2. RTL Inverter

MOSFET CHARACTERISTICS

VDS 3 0

VGS 2 0

M1 1 2 0 0 MOD1 L=4U W=6U AD=10P
AS=10P VIDS 3 1

- MODEL MOD1 NMOS VTO=-2
NSUB=1.0E15 UO=550
- DC VDS 0 10 .5
VGS 0 5 1
- END

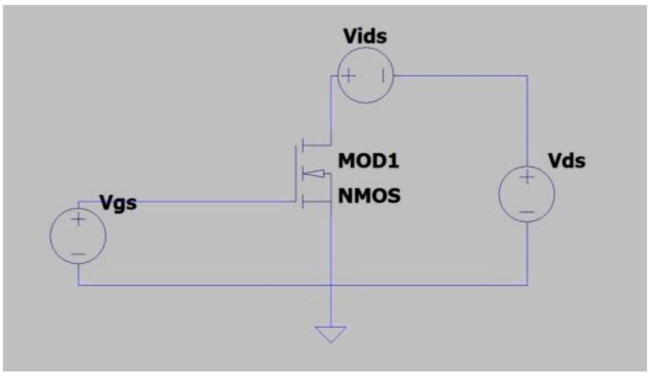

Fig3. MOSFET

IV. INTRODUCING RADIATION HARDENING TECHNIQUES

A. SEU INDUCED INVERTER CODE:

VCC 4 0 5

RB 1 2 10K Q1 3 2 0 Q1

RC 3 4 1K

.MODEL Q1 NPN BF 20 RB 100 TF .1NS CJC 2PF

SEU Event (Transient Pulse)

VIN_SEU 1 0 PULSE(0 5 0 2NS 2NS 30NS 1)

.DC VIN_SEU 0 5 0.1

.TRAN 1NS 100NS

.END

B. SEL INDUCED MOSFET

CHARACTERISTICS CODE: MOSFET Characterization Circuit with SEL Trigger

.OPTIONS NODE NOPAGE

Define the MOSFET Model

M1 1 2 0 0 MOD1 L=4U W=6U AD=10P AS=10P

.MODEL MOD1 NMOS VTO=-2 NSUB=1.0E15 UO=550

SEL Trigger (Simulated SEL Condition)

ISEL 2 0 PULSE(0 1E9 0 1NS 1NS 10NS 1)

DC Sweep for VDS

VDS 3 0

.DC VDS 0 10 .5

VGS 0 5 1

Measure Current (ID)

VIDS 3 1

.END*C. SEL INDUCED INVERTER CODE:*

VCC 4 0 5

RB 1 2 10K Q1 3 2 0 Q1

RC 3 4 1K

.MODEL Q1 NPN BF 20 RB 100 TF .1NS CJC 2PF

SEU Event (Transient Current Pulse)

I_SEU 1 0 PULSE(0 1U 0 2NS 2NS 30NS 1)

.DC I_SEU 0 1U 1U

.TRAN 1NS 100NS .

.END

V. RESULTS

A. SEU INDUCED INVERTER CODE:

The inverter code uses a periodic pulse waveform, while the SEU induced inverter code uses a single pulse. This will affect the output voltage waveform at the collector node 3. The periodic pulse waveform will produce a periodic output waveform with a phase shift and an amplitude reduction due to the transistor action. The single pulse will produce a single output pulse with a delay and a distortion due to the transistor action.

Vin:

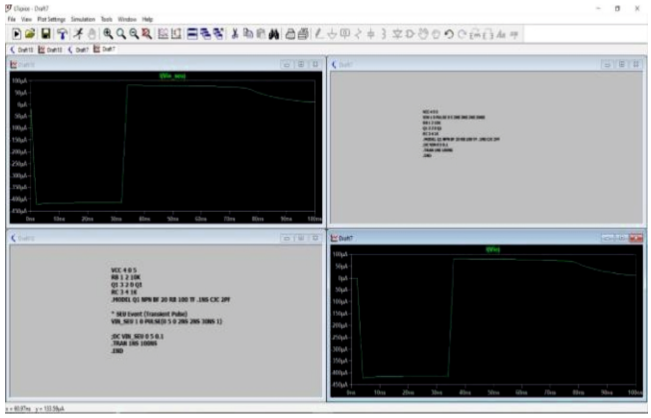

Fig4. SEL induced inverter with Vin variations

SEU induced pulse acting as a voltage source and switching on the device as input voltage, as shown above.

V(3):

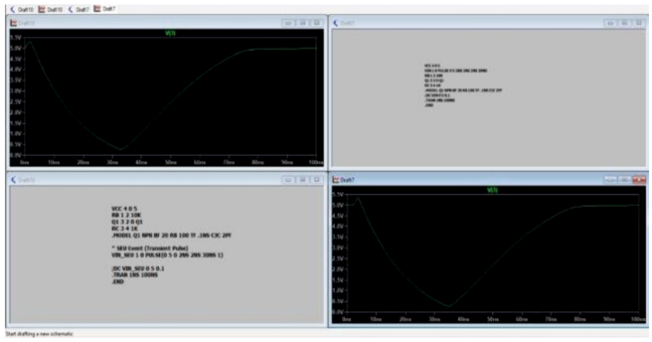

Fig5. SEL induced inverter with V(3) variations

Variation in the voltage V(3) due to SUE pulse driving the voltage pulse without a delay of 2ns.

B. SEL INDUCED INVERTER CODE:

If SEL induced inverter code is simulated, the simulator will plot the output current ID versus the drain-source voltage VDS for different values of the gate-source voltage VGS. The plot will show the MOSFET characteristics in different regions of operation, such as the cutoff region, the linear region, and the saturation region. The plot will also show the effect of the SEL pulse on the output current ID. The SEL pulse will cause a temporary increase in the output current ID due to the latch up phenomenon, which is a parasitic thyristor action that can damage the device. The SEL pulse will also affect the threshold voltage VTO of the MOSFET, which is the minimum voltage required to turn on the device. The SEL pulse will reduce the threshold voltage VTO, making the device more sensitive to the gate-source voltage VGS. This will change the output current ID for the same values of VDS and VGS. The SEL pulse will also introduce some noise and distortion in the output current ID waveform.

I (Rc):

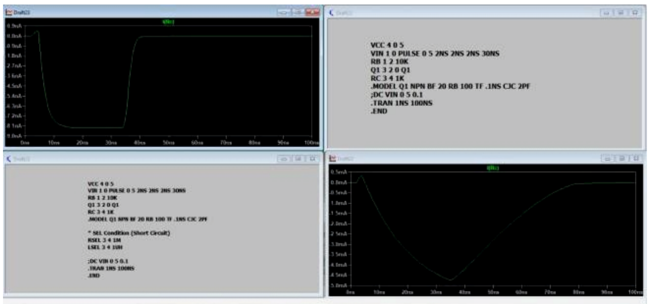

Fig6. SEL induced inverter with I(Rc) variations

Sudden spike in I(Rc) due to the SEL phenomenon causing sudden temporary raise in the output current that can damage the device.

Ic(Q1):

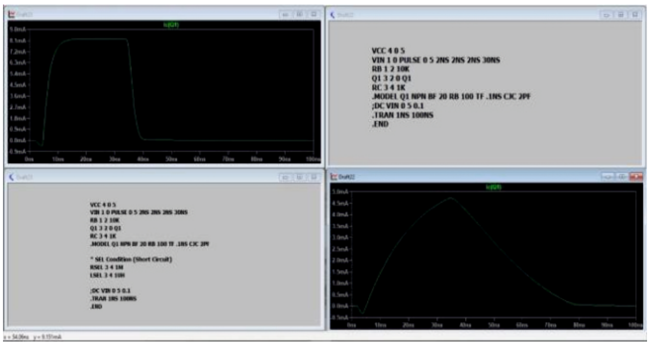

Fig7. SEL induced inverter with Ic(Q1) variations

The above result gives out the idea of how the current of the transistor Q1 raises abruptly due to the shorting effect in the device causing increasing of current temporarily.

V(1):

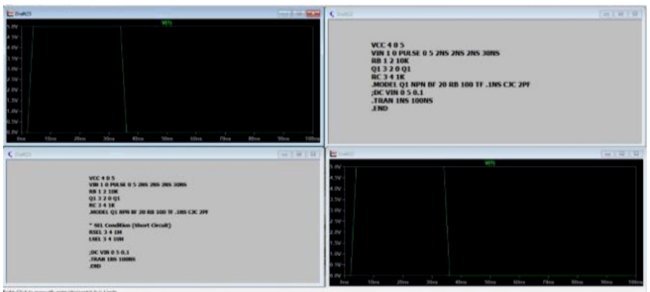

Fig8. SEL induced inverter with V(1) variations

The above result indicates that the SEL effect is only present in a specific region and does not affect or short the entire circuit. Furthermore, the result shows that voltage V(1) is functioning correctly with exact characteristics both with and without SEL induction.

C. SEL INDUCED MOSFET CHARACTERIZATION:

The SEL induced MOSFET code will plot the output voltage VCE versus the collector current IC for different values of the SEU pulse magnitude. The plot will show the BJT characteristics in different regions of operation, such as the active region, the saturation region, and the cutoff region. The plot will also show the effect of the SEU pulse on the output voltage VCE. The SEU pulse will cause a temporary increase in the base current IB due to the radiation strike, which will increase the collector current IC and reduce the output voltage VCE. The SEU pulse will also affect the BJT model parameters, such as the forward current gain BF, the base resistance RB, the forward transit time TF, and the collector-base junction capacitance CJC. These parameters affect the transistor characteristics, such as the current gain, the switching speed, and the frequency response.

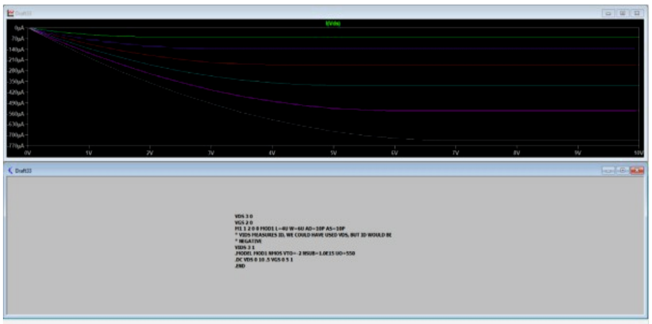

Fig9. SEL induced MOSFET with I(Vds) variation

The plot illustrates the change in Ids due to voltage pulse V(Ids). Vds was not used due to the result of negative Id. V(Ids) is the measuring unit for Ids.

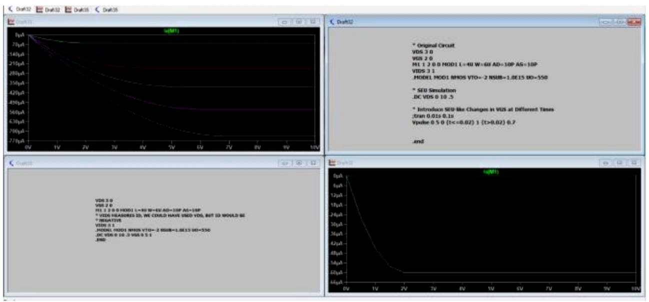

Fig10. SEL induced MOSFET with Is(M1) variation

The current shown with SEL effect in the above result, results in single current pulse.

VI. ADVANTAGES AND FUTURE SCOPE

A. ADVANTAGES:

Exploring Circuit Modeling and Radiation Effects with SPICE offers several advantages, both in terms of scientific understanding and practical applications.

Here are some of the key advantages:

Enhanced Radiation Resilience

Improved Safety in Critical Applications

Optimized Design

Cost Savings

Data Integrity

Scientific Advancements

Real-World Applications

Education and Training

Overall, the advantages of exploring circuit modeling and radiation effects with SPICE extend to improved safety, reliability, and innovation across a spectrum of applications, from space exploration to critical infrastructure and beyond. This research not only addresses current challenges but also lays the foundation for more robust and resilient electronic systems in the future.

B. FUTURE SCOPE:

The future scope of exploring circuit modeling and radiation effects with SPICE is promising and multifaceted, driven by ongoing advancements in electronics, space exploration, nuclear technology, and various other fields. Here are some key areas where this research can continue to evolve and make a significant impact:

Advanced Space Missions

Miniaturization and Nanoelectronics

Artificial Intelligence (AI) Integration

Terrestrial Applications

Materials and Component Development

Machine Learning-Assisted Design

Quantum Computing

Interdisciplinary Collaboration

In-Orbit Testing and Validation

Energy and Sustainability

The future scope of exploring circuit modeling and radiation effects with SPICE is intertwined with the evolving landscape of technology and scientific inquiry. As electronic systems become increasingly integral to our daily lives and to endeavors such as space exploration and critical infrastructure, research in this area will play a vital role in ensuring the reliability and safety of these systems in the face of ionizing radiation.

VII. CONCLUSION

The exploration of circuit modeling and radiation effects with SPICE is pivotal for various critical applications like space missions, nuclear technology, and medical devices. This research enhances our understanding of electronic systems and ionizing radiation, leading to advancements in radiation- hardened electronics and fostering innovation in safeguarding systems from radiation. Collaboration among experts drives progress, offering educational opportunities and shaping the future of advanced technology. This field ensures safety, sustainability, and reliability in electronic systems, shaping a technologically robust future.

REFERENCES

[1] Design of a 1 Gb Radiation Hardened NAND Flash Memory, Nikhil Gupta, Bert Vermeire, Hugh Barnaby.

[2] Investigation of Radiation Hardened TFET SRAM Cell for Mitigation of Single Event Upset M. POWN 1 AND B. LAKSHMI2 (Member, IEEE, 2020).

[3] A Survey of Radiation Hardened CMOS Techniques Srinitha Nimmakayala, Dept of Electrical and Computer Engineering Montana State University Bozeman, MT 59717 USA.

[4] Trade-off in Logical Radiation Hardening: Approach, Mechanisms, and Reliability Impacts Hycham Aboutaleb, ENSTA ParisTech Bruno Monsuez, PhD, ENSTA ParisTech. [5] SPICE Simulation of Radiation Induced Charges and Currents in Silicon Substrate, 2020 IEEE Latin America Electron Devices Conference (LAEDC) San José, Costa Rica.

[6] Predictive tools and Radiation-Hardening-by-Design (RHBD) techniques for SET and SEU in digital circuits, HAL.

***