

# TOPIC:- RF-SET Simulation Method

## Abstract

An efficient simulation method for transmission-type radio-frequency single-electron transistors (RF-SETs) is developed. By introducing equivalent circuits of propagating microwaves through RF cables, the master equation for RF-SETs is solved self-consistently using the conventional circuit simulator SPICE. This method includes the SET current model. By examining transmitted waves, the developed method is shown to successfully reproduce numerical reference methods. The method provides a simple and fast way to simulate RF-SETs, even with complicated circuit geometry and at high frequencies over GHz.

## Title

Simulation Method of Transmission-Type Radio-Frequency Single-Electron Transistor (RF-SET) by SPICE

## Introduction

The single-electron transistor (SET) is a highly sensitive electrometer used in measuring quantum states. However, its large output resistance and the typical cable capacitance limit its bandwidth to a few kHz. The radio-frequency SET (RF-SET) was developed to increase this bandwidth by using a tank (LC) circuit for impedance matching. While simulation methods for reflection-type RF-SETs have been reported, no such methods for transmission-type RF-SETs had been developed. This project introduces an efficient simulation method for the transmission-type RF-SET.

## Objectives

1. To develop an efficient simulation method for transmission-type RF-SETs.
2. To utilize the conventional circuit simulator SPICE by creating an equivalent circuit model.
3. To solve the master equation for the RF-SET self-consistently within SPICE.
4. To validate the developed method by comparing its results to numerical reference methods.

## Methodology

1. An equivalent circuit for the transmission-type RF-SET, including the SET current model, was introduced.
2. This model allows the master equation and differential equations for RF cable voltages to be solved self-consistently.
3. The SPICE circuit simulator is used to calculate all branch currents and node voltages.
4. The transmitted wave  $V_{t}(t)$  and reflected wave  $V_{out}(t)$  are then obtained from the simulated node voltages.
5. The simulation results are compared against a steady-state reference model to verify accuracy.

## Input parameter for analysis:

Add parameters for sine source v1

Enter offset value (Volts/Amps):
0

Enter amplitude (Volts/Amps):
0.4m

Enter frequency (Hz):
660M

Enter delay time (seconds):
0.1n

Enter damping factor (1/seconds):
0.1n

## Outcome

1. The developed SPICE method (using both steady-state and time-dependent SET models) successfully reproduces the numerical results of the reference method.
2. The method is shown to be accurate for simulating RF-SETs at relatively low-frequency regimes (e.g., ~660 MHz).
3. The method provides a simple, fast, and accurate way to simulate RF-SETs, even in complicated geometries and at high frequencies (over GHz).

## Output:

Expected Result:

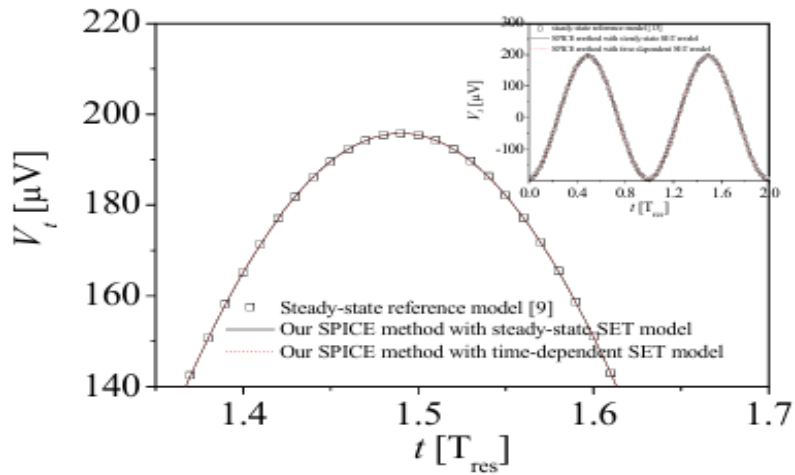
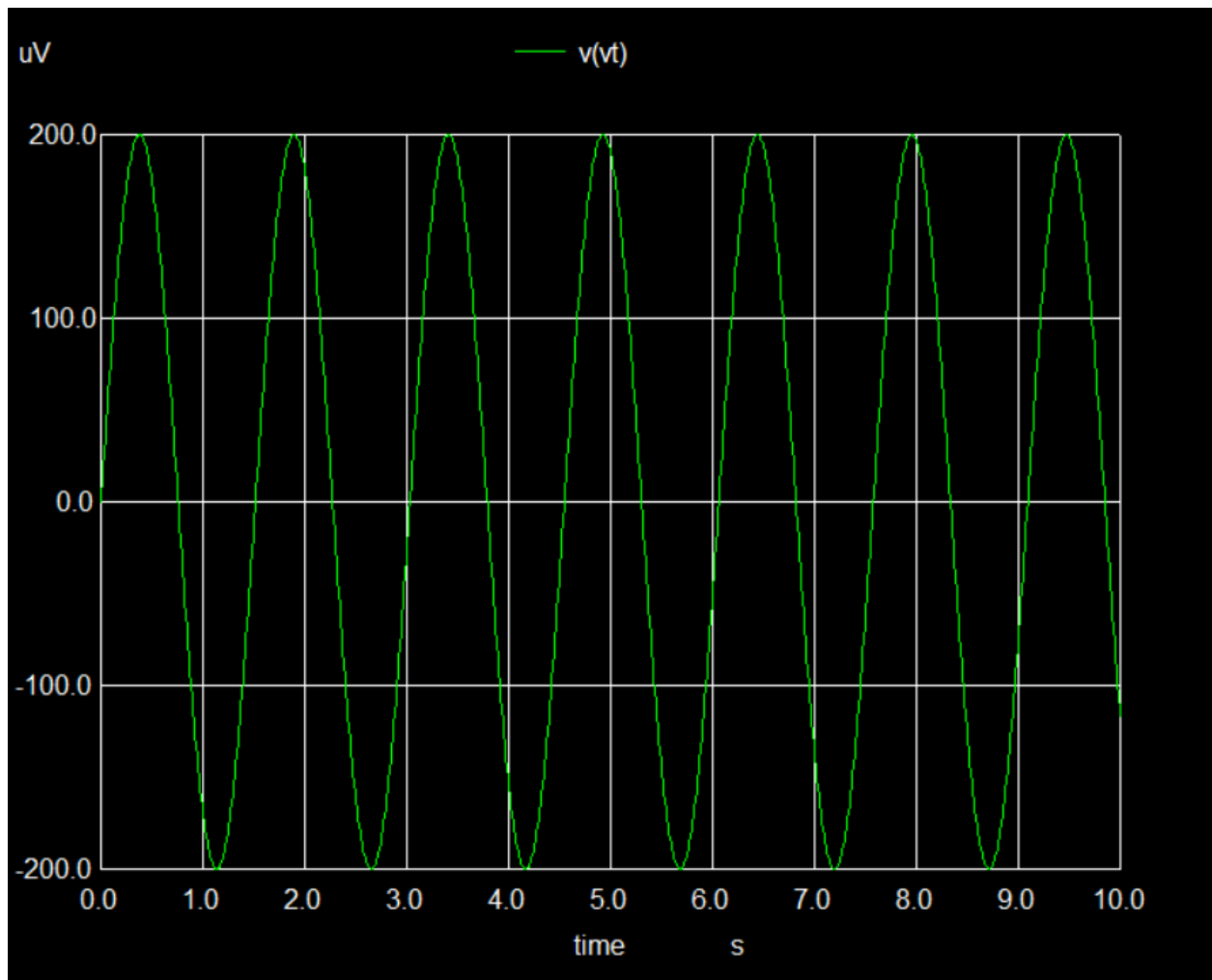


Fig. 2 Transmitted wave  $V_t(t)$  of the transmission-type RF-SET ( $C_d = C_s = 50$  aF,  $C_g = 20$  aF,  $R_d = R_s = 100$  k $\Omega$ ,  $T = 0.3$  K).

Observed Result:



### Applications

1. Designing and analyzing tank circuits for RF-SETs.
2. Analyzing the response and sensitivity of charges in quantum computation system.
3. Simulating high-frequency (GHz) circuits that incorporate SET devices.

### Conclusion

An efficient simulation method for a transmission-type RF-SET was developed. By introducing equivalent circuits, the master equation for the RF-SET was solved self-consistently using the SPICE simulator. The method was validated by showing that it successfully reproduces numerical reference methods. This provides a simple and fast way to simulate RF-SETs under complex conditions and at high frequencies.