Unconventional EDA for Mixed-Signal Circuits: A Graphene FET based LC-VCO Case Study

Shital Joshi
Computer Science and Engineering
University of North Texas, USA.
Email: ShitalJoshi@my.unt.edu

Saraju P. Mohanty
Computer Science and Engineering
University of North Texas, USA.
Email: saraju.mohanty@unt.edu

Elias Kougianos
Engineering Technology
University of North Texas. USA.
Email: elias.kougianos@unt.edu

Abstract

This paper presents ultra-fast non-EDA design and simulation of mixed-signal circuits/systems. Since SPICE simulators are slow and even computationally infeasible for large and complex circuits, this paper presents a complete Simscape® based design simulation where a Graphene Field Effect transistor (GFET) based cross coupled LC oscillator is used as a case study. The GFET behavioral model and its implementation are done in the Simscape® physical modeling language. The model is based on a drift-diffusion conduction mechanism of a dual-gate device and the kink region in the $I-V$ saturation characteristics is modeled via a displacement current. The results obtained from the Simscape® simulation match well with the results obtained from EDA based flows, such as SPICE, VHDL-AMS and other Analog Hardware Description Language (AHDL) models. To the best the authors' knowledge, this is the first ever paper presenting a Simscape® model of a GFET device and also performing design exploration of GFET based RF circuits using Simscape®.

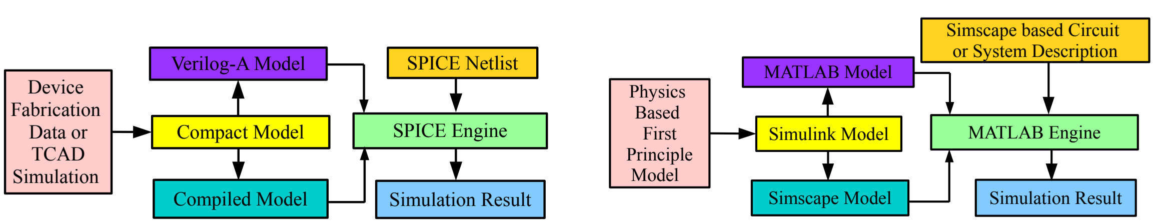
I. Introduction

In order to meet the demand for smaller, cheaper and low power consumption electronic products, designers have shifted their focus towards analog mixed signal systems on chip (AMS-SoC) [1] and alternative technologies. This paper considers the graphene FET to design nanoelectronic systems, due to the fact that graphene exhibits some exceptional properties: very high carrier mobility and current density, is immune to electromigration, has excellent thermal conductivity, mechanical toughness and flexibility and has fabrication process compatibility with silicon. As the level of integration of AMS increases, the cost and time-to-market also increases proportionally. In addition, SPICE simulation needs fab data or TCAD simulation to derive compact models. These pose a serious limit for emerging technologies where fab data are not available. This paper proposes a Simscape® based non-EDA design flow for integrated circuit (IC) design.

In this paper, a Simscape® behavioral model of a GFET based cross coupled LC oscillator is presented. The **novel contributions of this paper** to the state-of-art are the following: (1) A non-EDA paradigm shift to Simscape® for ultra-fast design exploration. (2) Modeling of a GFET device and case study circuit (LC-VCO) using Simscape® language. (3) Experimental validation of Simscape® language device level models with existing VHDL-AMS or Verilog-A models.

II. Proposed Unconventional Mixed-Signal Design Flow

As shown in Fig. 1(a), conventional EDA based design flows need fab data or process information in order to extract the compact model for the SPICE or analog simulation engine. Besides being effort intensive and computationally slow, these are not well suited to emerging technologies for which fab data are not yet available. However, the proposed Simscape® based non-EDA design flow (Fig. 2(b)) does not need any fab data but instead relies on first principle models published in the literatures. This enables the Simscape® based design to be easily applicable for emerging technologies. In addition, the availability of optimization options in MATLAB® makes it easier to fine tune the critical parameters, without additional computational overhead.



(a) SPICE Simulation Design Flow

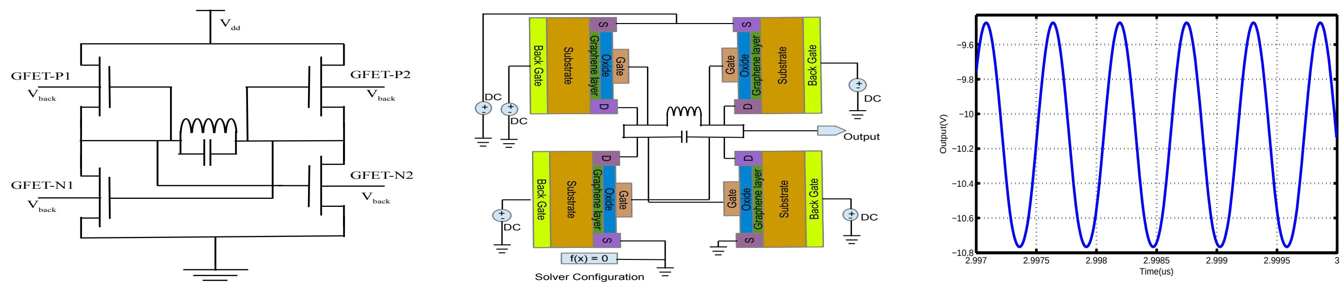
(b) Simscape Simulation Design Flow

Fig. 1. SPICE versus Simscape® based Design Simulation Flow.

Simscape® is a MATLAB® based object-oriented language which allows two ways of building a custom Simscape® model for emerging devices like GFET: (1) graphical method using fundamental Simulink®/Simscape® blocks, or (2) textually with the Simscape physical modeling language. This paper adopts the later approach due to its better portability. In order to compare the proposed Simscape® model with well verified EDA models, the paper considers the GFET model based on the VHDL-AMS model from [2] or Verilog-A model from [3].

III. Experimental Results and Conclusions

Due to high carrier mobility and high intrinsic voltage gain, GFETs can be used for very high speed communication components. As a case study, a cross-coupled oscillator is considered in this paper, as shown in Fig. 2(a). By applying appropriate top and back gate voltages, the threshold voltage can be controlled. To form an n-type channel, the top gate voltage is biased positively as compared to the threshold voltage and the reverse in the case of p-type channel.



(a) Schematic of LC oscillator using GFET

(b) Simscape® Model of Graphene based Oscillator

(c) Simscape® Simulation of the Graphene Oscillator

Fig. 2. GFET based LC oscillator configuration and the output oscillation

In order to start the oscillation, the transconductance of the active device should follow the following equation:

$$g_{active} \geq \frac{RC}{L}, \tag{1}$$

where R is the resistive loss of the tank due to the parasitic resistance of inductor.

The transconductance of the active device is controlled by modifying the width W while keeping the channel length of both PFET and NFET fixed. In the current limited region, the output voltage swing is defined by:

$$V_{tank} = \frac{I_{bias}}{g_{tank}}, \tag{2}$$

where I_{bias} denotes the bias current and g_{tank} is the total tank conductance. When the oscillator enters the voltage limited region, output voltage amplitude is limited by supply voltage and the operating region of active device.

To configure the n-channel transistor, the critical electric field (E_c) is set to 15 KV/m. The back gate voltage and width of the channel are characterized to obtain the desired bias point.

Fig. 2(b) shows the Simscape® based simulation setup. In order to perform accurate simulation, the solver used in this paper is `ODE14X` (Extrapolation), which has the minimum possible step size. The PFETs are arranged in a cross coupled topology and are characterized to operate close to the saturation region. In order to obtain over unity intrinsic gain, the source to drain conductances are reduced. The oscillator is designed to operate at 1.8 GHz having tank voltage swing 1.286 V_{p-p}. Fig. 2(c) shows the final output obtained from the simulation.

IV. Conclusions and Future Research

A Simscape® based behavioral model, suitable for design exploration at high levels of abstraction, has been presented in this paper and the results are compared with well-accepted EDA models. The results showed that the Simscape® based model can be used as a substitute for more detailed but time consuming EDA models and it provides RF designers with a unique design exploration and verification tool.

As a future research, additional functionalities for noise, transfer function and non-linear RF analyses such as periodic and quasi-periodic steady state can be incorporated within the Simscape® model. Optimization techniques using particle swarm-based optimization (PSO) algorithms such as artificial bee colony and ant colony optimization for GFET based circuits will be explored within Simscape®.

References

[1] S. P. Mohanty, *Nanoelectronic Mixed-Signal System Design*. McGraw-Hill Education, 2015.

[2] I. J. Umoh and T. J. Kazmierski, "VHDL-AMS Model of A Dual Gate Graphene FET," in *Proceedings of the Forum on Specification and Design Languages*, 2011, pp. 1–5.

[3] M. A. Khan, S. P. Mohanty, and E. Kougianos, "Statistical Process Variation Analysis of a Graphene FET based LC-VCO for WLAN Applications," in *Proceedings of the 15th IEEE International Symposium on Quality Electronic Design (ISQED)*, 2014, pp. 569–574.