OptSim Circuit:

A SPICE-Like Photonic Simulator for Scalable Photonic Integrated Circuits

October 2017

Author

Jigesh Patel
Senior Application Engineer

Introduction

The time for large-scale, reliable photonic integrated circuits (PICs) has finally arrived. Governments, indispensable to PICs as was (and still is) the role of electronic design automation (EDA) tools to the fruition of Moore's law for CMOS over the last 50 years. This document examines fundamental differences in numerical modeling of PICs compared to the electronic counterpart and explains the rationale behind the simulation approach OptSim Circuit has to offer.

Simulation of Electrical Circuits

SPICE and SPICE-like simulators have been widely used since the 1970's for time- and frequency-
inr anC9eid s c8 (s 0 T6 n)--]3 c8 1 (r 4.5 (s h8a)p)-12.5 (1 (r a)-12.86 Lang (en-US)7 8a)(C97.2(r as217.7 (u5 (c)-c-9.2 4

A compact model can be based on analytical solution of the underlying physics (for example, rate equation-based laser model or analytical solution of a ring modulator), or it can be based on frequency response

Modeling custom PIC element using OptSim Circuit cosimulation with MATLAB (https://www.synopsys.com/optical-solutions/rsoft/rsoft-product-applications/modeling-custom-pic-elements.html): As the push towards photonic integration intensifies, designers want flexibility to develop proprietary intellectual property (IP) and model custom PIC elements via cosimulation with the third-party tools such as MATLAB. With OptSim Circuit, it is possible to use cosimulation with MATLAB to model custom bidirectional PICs and PIC elements. As an illustration, this case study depicts a notch filter PIC [Ref. 5] with a custom waveguide modeled in MATLAB

From ideas to photonic chip (https://www.synopsys.com/content/dam/synopsys/optical/pdf/RSoft-OptSim-Circuit_Interface_IPKISS.pdf): This case study illustrates the PIC design flow from ideas to OptSim Circuit schematic-based simulation and generation of mask via interface to third-party tools

Summary

In this white paper, we discussed the OptSim Circuit simulation philosophy and its rationale. This approach is scalable with component count, complexity of design and layers of integration. The OptSim Circuit simulation is substrate technology-neutral; whether it is CMOS-based silicon photonics, lithium niobate, or III-V material, the photonic simulation and design flow is the same. Tight integration with OptSim makes it convenient to test and optimize PICs from the system perspective; cosimulation with MATLAB offers virtually limitless expansion possibilities. Ever-growing support for foundries and their PDKs—together with interfaces to mask layout tools—make OptSim Circuit an unparalleled platform for migrating from ideas to fabrication.

Acknowledgement

The author is grateful to Enrico Ghillino, Dr. Tom Walker, Dr. Pablo Mena, and Dr. Evan Heller of Synopsys Optical Solutions Group, and Dr. Dwight Richards of College of Staten Island, City University of New York (CUNY) for stimulating discussions and feedbacks.

To Learn More

At (em Ce8at.m Ce8at.

Synopsys, Inc. · 690 East Middlefield Road · Mountain View, CA 94043 · www.synopsys.com

10/11/17.RD_CS11910_RSoft OptSim_C_SPICE_like Photo_Sim_WP.