

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, academia and industry are all onboard to transfer the knowledge gained from decades of manufacturing experience in electronic integrated circuits to photonic integration. In addition to the favorable economics from higher energy efficiency, smaller footprint and lower cost, photonic integration is recognized as essential to national security. The role of photonic design automation (PDA) tools is going to be as 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-domain analyses of linear and nonlinear analog electrical circuits. The circuits are made up by connecting fundamental electrical components such as transistors, diodes, resistors, inductors, capacitors, and power sources. Certain optoelectronic devices can also be simulated using equivalent circuit-level representations. For example, [Ref. 1] describes a model for vertical cavity surface emitting lasers (VCSELs) that can be simulated in conjunction with electronic circuitry.

Simulation of Photonic Circuits

Photonic circuits create closed propagation paths either due to the circuit itself (such as coupled ring resonators and modulators as shown in Figure 1) or through multiple orders of reflections and multipath interference (MPI) from neighboring components (as shown in Figure 2). This is in contrast with long-distance fiber-optic data communication links where optical signal propagates in one direction, from transmitter to the receiver, with negligible reflections. Performance of a photonic circuit is influenced by sourcing and sinking from preceding and proceeding elements, as well as concentrated (lumped) and distributed (traveling-wave) interactions. However, unlike real-valued electrical signals in SPICE, the complex nature of optical signals requires photonic circuit simulators to account for the state of polarizations, relative and absolute phases, transverse mode profiles and wavelengths of light.

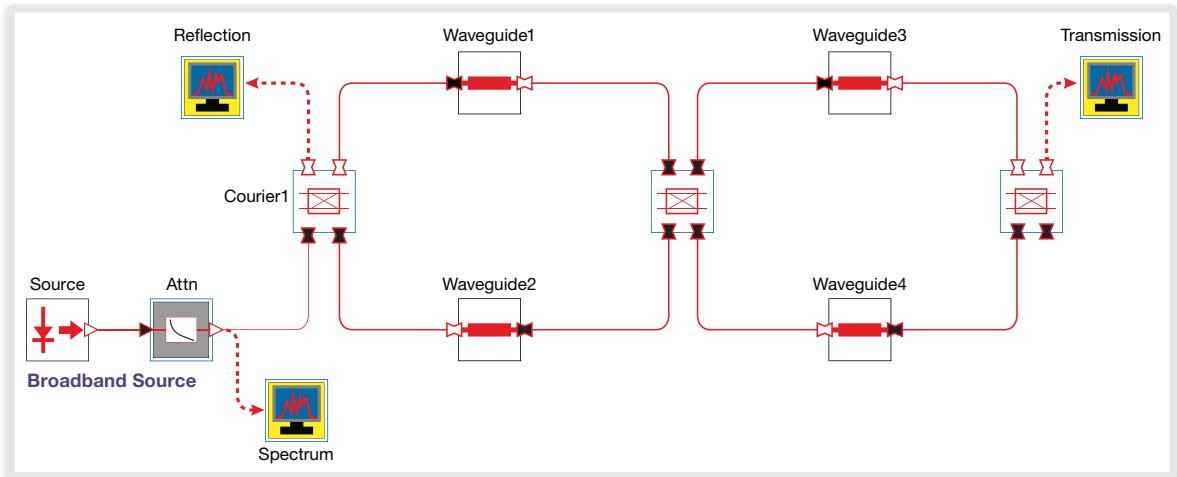


Figure 1. Photonic circuit schematic formed by coupled optical resonators



Figure 2. Illustration of a photonic circuit formed by multipath interference from neighboring PIC elements

Photonic Device Versus System Approach for Simulation

Most photonic device simulators use different methods to solve the Maxwell's electromagnetic wave equations depending on the nature of the device, its size and geometry. Since a PIC can be made up of many active and passive electrical and photonic components of different materials and dimensions with multi-directional, time-varying signal flows, a one-size-fits-all solution does not exist. Moreover, an à la carte approach based on device-dependent modeling algorithms is neither seamless nor efficient. As a result, a SPICE-like photonic systems modeling approach presents itself as a natural choice.

The OptSim Circuit Approach

OptSim Circuit is a SPICE-like photonic simulator for time- and frequency-domain, linear and nonlinear photonic circuit simulations where each device is represented as a compact model and a PIC is made up of several compact models, as shown in Figure 3.

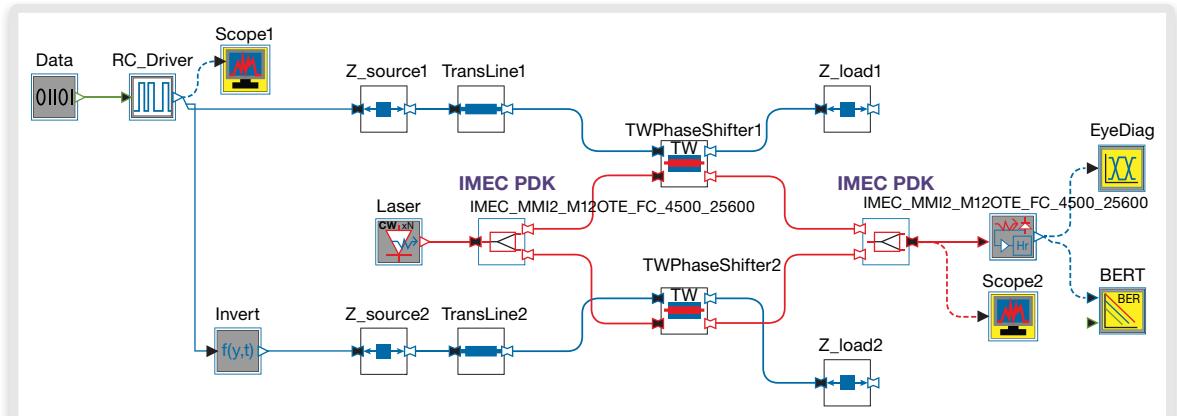


Figure 3. A Travelling-Wave Mach-Zehnder (TW-MZM) PIC schematic in OptSim Circuit using compact models of electrical, photonic and PDK components

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 characterization (either measured or from device modeling tools, either transfer matrix or scatter matrix), or it can be a process design kit (PDK) element from a foundry, which may have been extracted from physical modeling and attuned to experimental data from the wafer runs (for example, a splitter PDK with measured loss and reflection in each arm that can be attributed to the fabrication process).

Benefits of OptSim Circuit

The choice of the right simulation methodology has a direct impact on innovation, productivity, efficiency, and ultimately the bottom line of the business. Here's a high-level summary of some of the benefits OptSim Circuit brings to your organization:

- ▶ Accuracy
Support for bidirectional signal propagation naturally accounts for multiple orders of reflections and MPI with arbitrary delays and phase-shifts, transmission line travelling wave effects due to feeds, terminations and velocity mismatches without approximations.
- ▶ Speed and efficiency
Compact model characterization of circuit elements brings speed and efficiency to time-varying, linear and nonlinear PIC simulations that otherwise are intractable via device-level electromagnetic wave propagation methods.
- ▶ Flexibility and ease of use
Support for MATLAB co-simulation opens up virtually unlimited possibilities for adding and expanding component libraries. Support for foundry PDKs and interfaces with third-party mask layout tools provide a comprehensive pathway from ideas to fabrication. Seamless integration with OptSim makes it very convenient to test and optimize the PIC performance in the context of deploying it in a system. In addition, support for hierarchical components makes PICs scalable as demand, component count and complexities increase.

Suggested PIC Design Case Studies for Further Reading

For the sake of brevity, we have chosen to limit the scope of this white paper to the discussion of the simulation philosophy of OptSim Circuit and not include PIC design case studies. However, if you are interested in PIC design case studies, following is a list of suggested additional resources available on the Synopsys Optical Solutions Group website. In addition, please contact us at optics@synopsys.com if you have any questions or custom design requirements you would like to discuss.

- ▶ Performance of silicon photonic ring modulator-based transceivers in coherent fiber-optic systems (<https://www.synopsys.com/optical-solutions/rsoft/rsoft-product-applications/optsim-circuit-silicon-photonic-ring.html>): This case study illustrates simulation of a silicon photonic PIC and analysis of its performance at the system level [Ref. 2]. Performance analysis for the PIC is carried out for 32- and 16-Gbaud polarization multiplexed quadrature phase shift keying (PM-QPSK). Comparison is made in terms of optical-signal-to-noise-ratio (OSNR) penalty for transceivers based on lithium niobate (LiNbO₃) and silicon photonic (SiP) microring modulators
- ▶ Estimating the impact of wafer-to-wafer and run-to-run foundry process variations on the yield of a travelling-wave Mach-Zehnder Modulator (TW-MZM) chip (<https://www.synopsys.com/optical-solutions/rsoft/rsoft-product-applications/estimating-impact-wtw-rtr.html>): This case study demonstrates how to model a TW-MZM using discrete PIC elements. We replace the generic splitter and combiner model with IMEC PDK elements and study the influence of variations in fabrication processes [Ref. 3]
- ▶ Modeling coupled-resonator optical waveguide (CROW) (<https://www.synopsys.com/optical-solutions/rsoft/rsoft-product-applications/modeling-crow.html>): A coupled-resonator optical waveguide (CROW) device consists of a chain of coupled resonators in which light propagates with characteristics similar to a waveguide [Ref. 4]. The application spectrum for CROW devices is interesting and broad, ranging from slow light and sensors to waveguides with controlled dispersion and bandwidth. OptSim Circuit's ability to model feedback and bi-directional signal propagation makes it ideal for CROW analysis

- ▶ 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 (<https://www.synopsys.com/optical-solutions/rsoft/rsoft-system.html>) , you can find detailed product information, application notes, e-newsletters, and product literature. You can also contact us at optics@synopsys.com to request more information and a 30-day free trial of our software solutions.

Reference

- [1] Mena, P. V., et al., “A comprehensive circuit-level model of vertical-cavity surface-emitting lasers,” Journal of Lightwave Technology, vol. 17, no. 12, pp. 2612-2632, December 1999.
- [2] Ghillino, E., et al., “Simulation of silicon photonic coherent PM-QPSK transceivers using microring modulators,” 16th International Conference on Transparent Optical Networks (ICTON), July 6-10, 2014, Graz, Austria.
- [3] Patel, J., et al., “Silicon photonic foundry processes and travelling-wave Mach-Zehnder modulators,” Novus Light Technologies Today, Nov. 2015, http://www.novuslight.com/silicon-photonic-foundry-processes-and-traveling-wave-mach-zehnder-modulators_N4838.html
- [4] Poon, J. K. S., et al., “Designing coupled-resonator optical waveguide delay lines,” Journal of Optical Society of America (JOSA) B, Vol. 21, No. 9, pp. 1665-1673, 2004.
- [5] Lumeau, J., et al., “Phase-shifted volume Bragg gratings in photo-thermo-refractive glass,” Optical Components and Materials V, Proceedings of SPIE, vol. 6890, 2008.