



Press Contact  
Richard Rambaran  
Optiwave Systems Inc.  
613-224-4700 x303  
[media@optiwave.com](mailto:media@optiwave.com)

**FOR IMMEDIATE RELEASE**

## **Optiwave Unveils Industry's First Opto-Electronic Circuit Design Software**

---

### **OptiSPICE 1.0 Addresses Increasing Need for Design and Verification of Opto-Electronic Integrated Circuits.**

**OTTAWA, ONTARIO — September 15, 2009** — Optiwave Systems Inc., the leading provider of optical component and system design tools, today announced the immediate availability of OptiSPICE 1.0, the industry's first circuit design software for analysis of integrated circuits including interactions of optical and electronic components. OptiSPICE is the only offering that enables the design and simulation of opto-electronic circuits at the transistor level, from laser drivers to transimpedance amplifiers, optical interconnects and electronic equalizers.

“OptiSPICE is the next evolutionary step in advanced simulation design software created to meet the current demands of today's electronic and optical engineers,” said Dr. Jan Jakubczyk, President of Optiwave. “With the imminent coexistence of electrical and optical components at the chip and board level, it is important to provide designers with a reliable simulation framework that can accurately and efficiently predict signal behaviour in opto-electronic integrated circuits.”

Simulation of optical and electrical components simultaneously is a multi-disciplinary problem. Equations governing optical and electrical components are incompatible for co-simulation within traditional numerical engines. In addition, optoelectronic components are temperature dependent and this dependence needs to be incorporated into the simulation to obtain correct results.

“Developing a revolutionary methodology to model optical signals in an electrical simulation framework suitable for ensuring reliable and self-consistent simulation of optical and electrical signals formulated for incorporation in a single-engine is what makes OptiSPICE a pioneering effort,” said Dr. Pavan Gunupudi, Professor at Carleton University. “Engineers now have one ideal modeling solution to meet their complex design requirements.”

[ MORE ]

OptiSPICE is the only circuit design software for the self-consistent simulation of optical, electrical, and thermal energy domains. Supporting a wide variety of electrical circuit elements such as diodes, transistors, BJTs and MOSFETS along with optical components such as laser diodes, optical fibers and photodiodes; OptiSPICE provides transient time domain, small-signal frequency, and noise analysis.

“With OptiSPICE as the foundation of our opto-electronic circuit design software initiative, we’re now in a unique position to deliver a unified platform that enables the world’s top research and development labs to make effective and confident decisions at all stages of the development lifecycle,” said Dr. Jackson Klein, Director of Optical Systems at Optiwave. “We are proud to have this exclusive product offering available to bridge the needs of researchers in both the electronic and optical domains.”

#### **About Optiwave Systems Inc.**

Optiwave is the leading provider of essential design tools for photonics, hosting an unparalleled suite of award-winning simulation software products. Optiwave provides a comprehensive variety of optical component and system design tools to hundreds of leading high-technology businesses and academic institutions. An established community of design engineers and scientific users in over fifty countries worldwide supports Optiwave’s hallmark of achievements in Canadian business for over a decade.

Optiwave is headquartered in Ottawa, Canada with an established distribution network throughout the Americas, Europe, and Asia. To download a free evaluation license, please visit Optiwave’s online resource centre at [optiwave.com](http://optiwave.com).

###