Optiwave

OptiSPICE

The First Opto-Electronic Circuit Design Software

Optiwave

7 Capella Court, Ottawa,
ON, Canada, K2E 7X1

Phone: 1-613-224-4700
Toll free number: 1-866-576-6784

www.optiwave.com

Overview

OptiSPICE is the first circuit design software for analysis of integrated circuits including interactions of optical and electronic components. It allows for the design and simulation of opto-electronic circuits at the transistor level, from laser drivers to transimpedance amplifiers, optical interconnects and electronic equalizers. OptiSPICE produces self-consistent solutions of opto-electronic circuits that contain feedback spanning both optical and electrical parts. OptiSPICE is a fully-integrated solution for parameter extraction, schematic capture, circuit simulation and waveform analysis.

Description

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 and boards.

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. Temperature effects are particularly important for the modeling of optical devices integrated on a common substrate with high power electrical components such as laser drivers. In this case the close proximity of the electronics and optical elements such as lasers produces significant thermal coupling. The coupling of the electrical, optical and thermal domains in such a situation requires a self-consistent solution.

OptiSPICE is the only circuit design software for the self-consistent simulation of optical, electrical and thermal energy domains. The optical elements are represented by delay-differential equations, the electrical circuit by algebraic differential equations and the thermal circuit by a set of first order nonlinear heat diffusion equations. 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.

Specific Benefits

- Significantly reduce product development costs and boost productivity through OptiSPICE comprehensive design environment to simulate optical and electrical circuits in one simulation engine.
- Run state-of-the-art transient time domain, small-signal frequency, and noise analysis to accurately predict behaviour of advanced opto-electronic circuits.
- OptiSPICE Schematics offers direct schematic entry in an intuitive graphical user interface. It allows for greater ease of schematic capture, parameter specification, waveform probing and usage.
- Waveform analysis using OptiSystem for complex post-processing functionality. Advanced visualization tools produce OSA Spectra, signal chirp, eye diagrams, polarization state, constellation diagrams and much more.
- Includes parameter extraction tools for OptiSPICE model creation. From measurement data, parameter extractors are used to find the best set of OptiSPICE model parameters to fit the measurement.

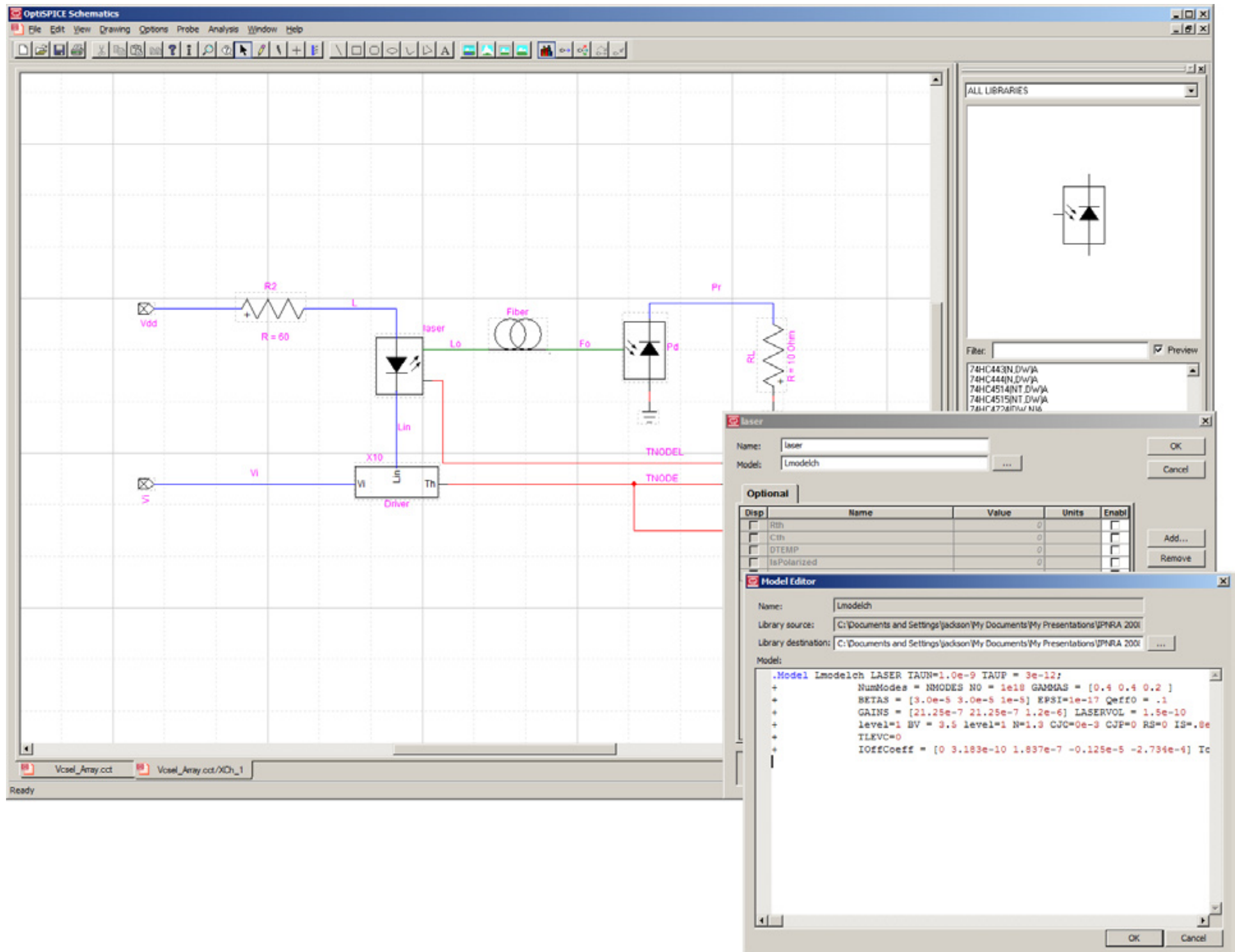
Applications

- Design and simulation of opto-electronic circuits at the transistor level, from laser drivers to transimpedance amplifiers, optical interconnects and electronic equalizers.
- Signal integrity analysis of opto-electronic circuits, including eye diagram analysis with BER patterns.

Key Features:

SCHEMATIC EDITOR:

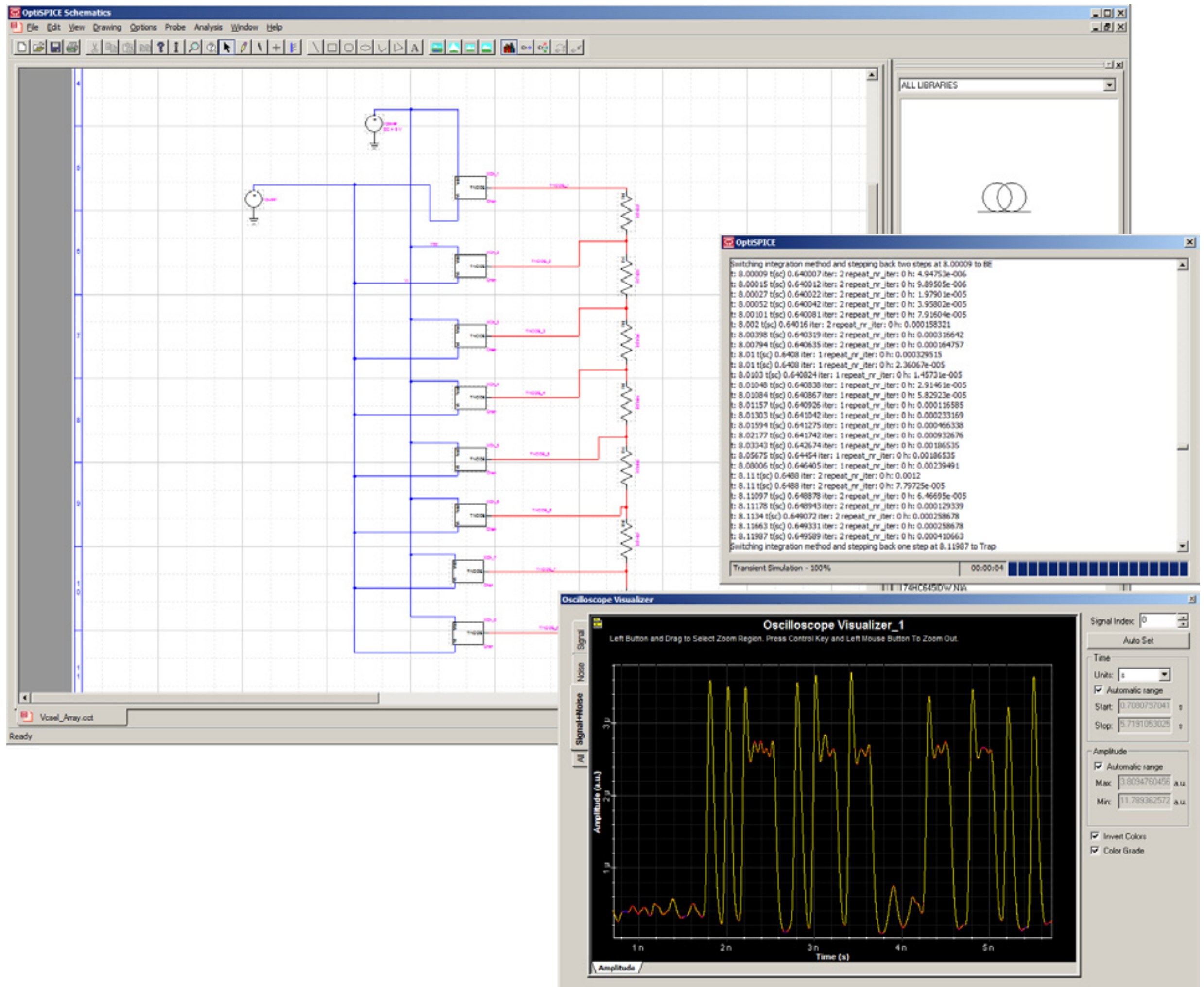
- Integrated Device Symbol Editor allows you to create custom symbols for devices or hierarchical blocks using standard drawing tools.
- Hierarchical Design with unlimited levels is fully supported. Any symbol on a schematic can contain another schematic of arbitrary size. Blocks can be nested to any desired depth. Any number of hierarchical blocks can be open for editing at any time.
- OptiSPICE Schematics includes a powerful Custom Report Generator tool for netlist and text report generation. The report format is driven by a "form file" which contains formatting commands and constant text. Form file features allow you to control: Overall report structure, e.g. netlist formats by signal or by device, listings by device for bills of materials, etc.
- OptiSPICE includes several powerful technologies for scripting and customization that allows full access to all design data and virtually every program function.
- The schematic editor can save diagrams in the standard PDF (Acrobat), WMF (Windows Metafile) and DXF (AutoCAD) graphics formats. This capability allows you to pass graphics to other programs for plotting, enhancement, or incorporation into other documentation.
- Generate OptiSPICE or HSPICE compatible netlists.



Key Features:

SIMULATOR

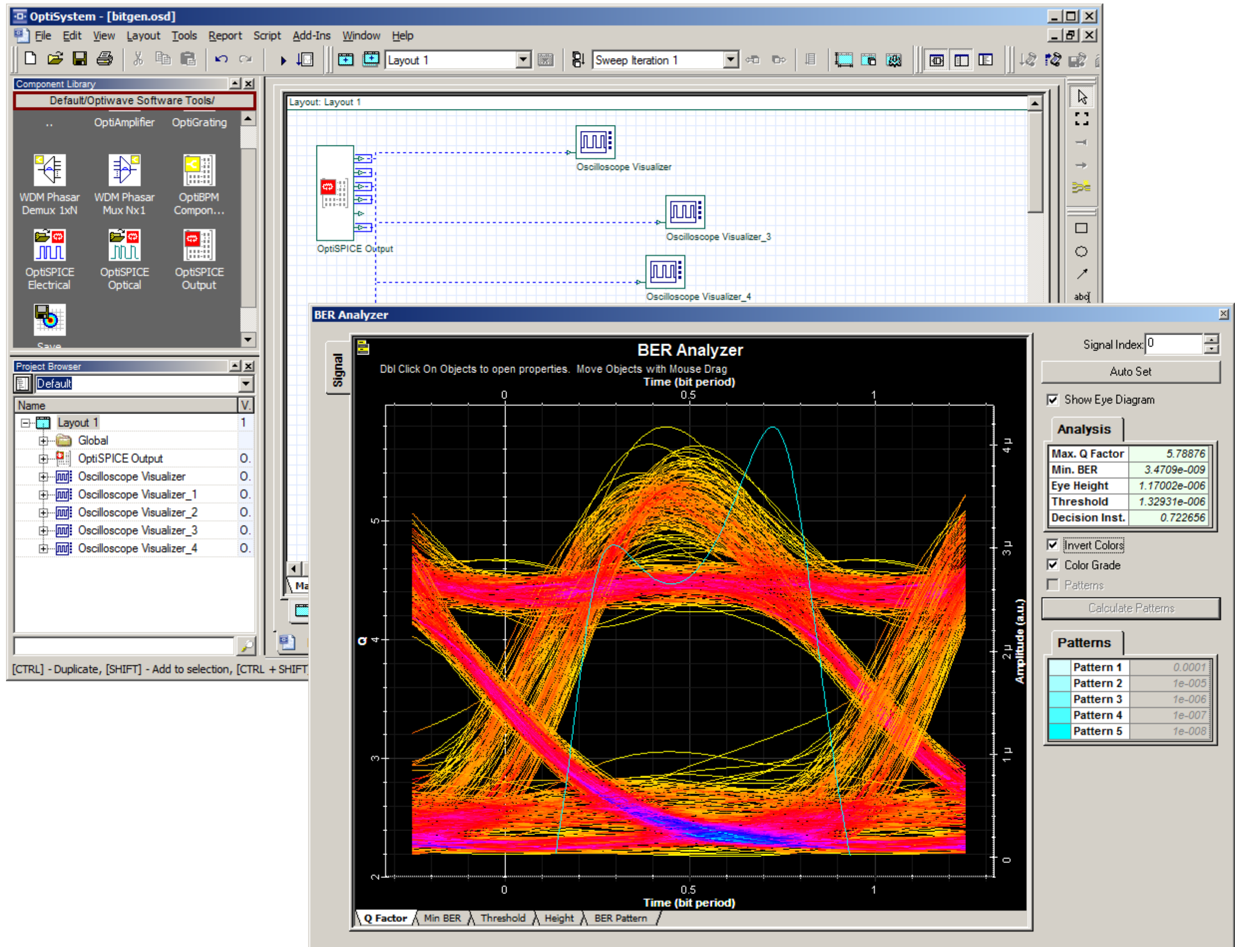
- OptiSPICE simulator incorporates equations governing optical components directly into an electrical simulation framework, thus forming a single-engine opto-electronic simulation software.
- Includes thermal macro models that model the thermal behaviour of the devices. Users can incorporate them into the opto-electronic simulation to provide reliable simulation results.
- Supports 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.
- Able to handle integrated optics, multiple optical channels (WDM), and multimode signals.
- Advanced numerical techniques for superior convergence. Advanced solver automatically selects the best convergence algorithm for reliable transient simulation convergence.
- Active and passive device model compatibility with industry HSPICE standard. Users can easily import external models and netlists written in HSPICE format to OptiSPICE.
- Enables accurate simulations by supporting BSIM3 models.
- Provides accurate implementation of different frequency dependent models including S-parameters, pole/residue expressions and transmission line models.



Key Features:

WAVEFORM ANALYSIS:

- OptiSPICE includes the award winning software OptiSystem for complex post-processing, signal integrity and waveform analysis.
- In order to predict the system performance, OptiSystem calculates parameters such as BER and Q-Factor using numerical analysis or semi-analytical techniques for systems limited by inter symbol interference and noise.
- The OptiSystem signal processing library includes many components for post-processing of OptiSPICE results. Users can also create new types of analysis or incorporate new components using MATLAB.
- Advanced visualization tools produce OSA Spectra, signal chirp, eye diagrams, polarization state, constellation diagrams and much more. Also included are WDM analysis tools listing signal power, gain, noise figure, and OSNR per channel.

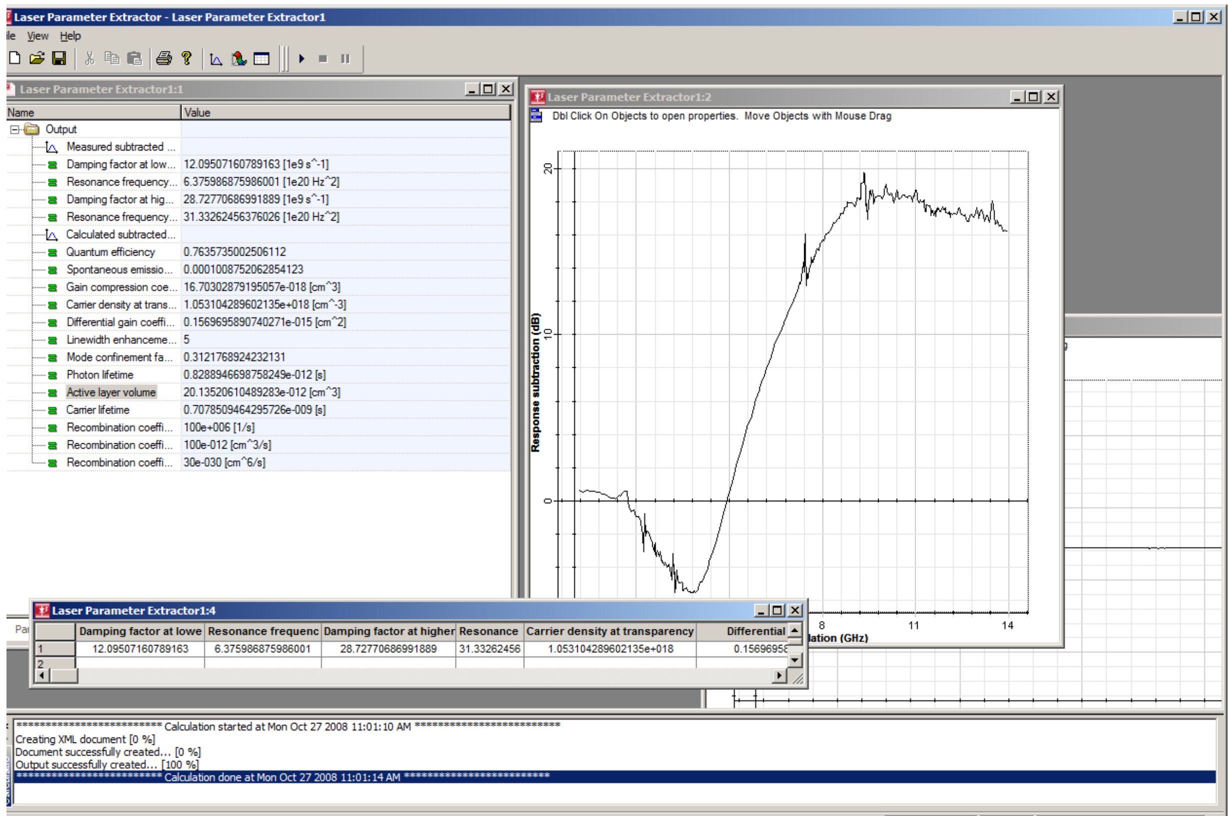


Key Features:

PARAMETER EXTRACTION:

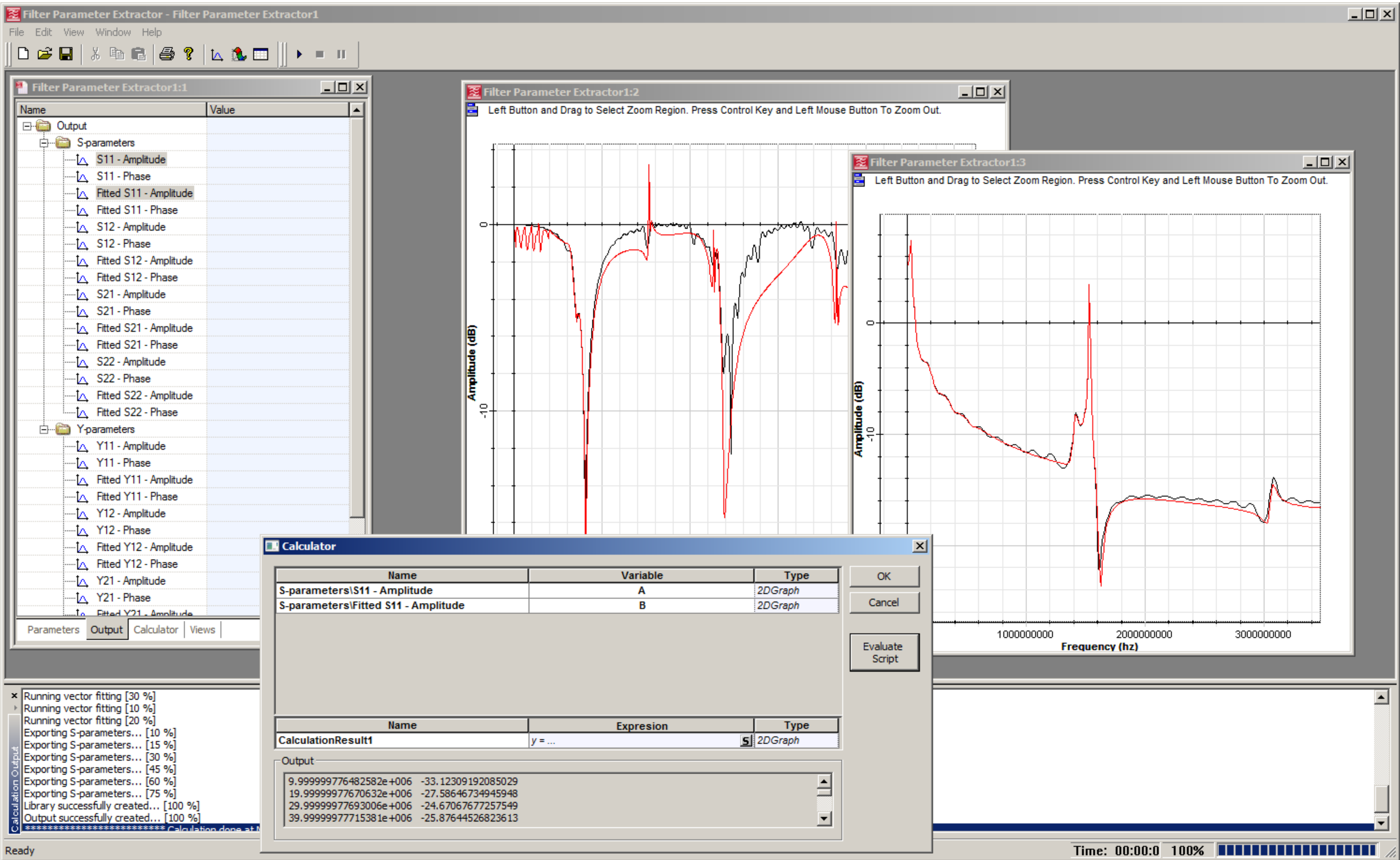
- Laser parameter extractor allows users to generate models by extracting and fitting parameters from static and dynamic measurements of lasers.
- Filter parameter extractor allows users to translate S-parameters into compact and efficient pole/residue representations.
- Multimode fiber parameter extractor includes an optical fiber mode solver that allows users to generate libraries of fibers from a user defined refractive index profile.

Laser Parameter Extraction:



Key Features:

Filter Parameter Extraction:



Key Features:

Fiber Parameter Extraction: