



OptiSPICE

The First Opto-Electronic Circuit
Design Software

4.0

New Features

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.



Key Features for OptiSPICE 4.0

OptiSPICE 4.0 introduces a new anchor feature, Waveform Viewer; a post processing analysis tool that will provide users with a means to simultaneously view electrical and optical signal data captured from all probes placed in an OptiSPICE circuit design.

With this new visualization capability it will now be possible to analyze simulation results directly within OptiSPICE, or alternatively through OptiSystem, all through a simple click of a button! Users will thus be able to deploy OptiSPICE for end-to-end analysis of opto-electronic circuit designs and, as required, take advantage of OptiSystem's advanced post-processing visualizers for additional analysis.

Also included in OptiSPICE 4.0 is a new installer for OptiSPICE samples. With this new feature, users will be able to quickly setup their preferred location for the OptiSPICE samples directory. The installer can be used as many times as required, so it is possible to install multiple instances of the samples directory in different folder locations!

OptiSPICE Waveform Viewer: Overview

The Waveform Viewer is post processing analysis tool that allows users to view the data captured from all probes placed in an OptiSPICE circuit design.

It can be used to review the data captured following Transient, AC, DC and Operating point analysis of electrical, optical and opto-electronic circuits. Access to the Waveform Viewer is automatically displayed following completion of an Analysis > Run. An example output from Waveform Viewer is shown in Figure 1.

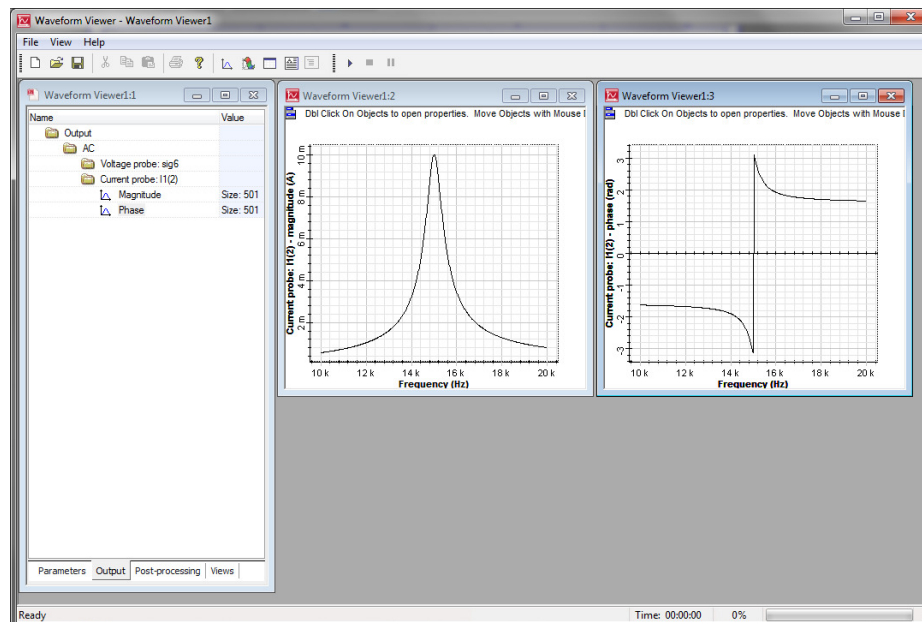


Fig 1: OptiSPICE Waveform Viewer. Data from all probes in an OptiSPICE circuit design are available via a menu tree structure with the “Output” tab. Double-clicking on any of the 2D icons opens a signal waveform plot for the probe.

OptiSPICE Waveform Viewer: Main features

The main features of the Waveform Viewer include:

Post-processing data visualization

Following an Analysis > Run all probe locations can be quickly accessed (via a menu tree architecture) and viewed in a data grid, text or 2D graph format. Multiple views can be opened at once (See Figure 2).

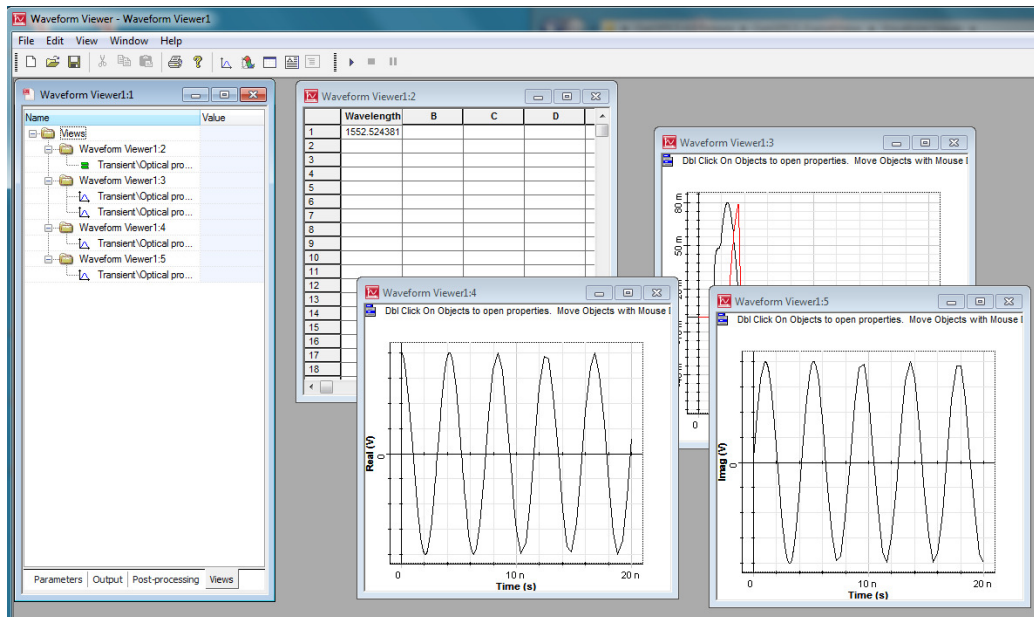


Fig 2: Multiple views of parameters and data probes can be analyzed simultaneously. 2D graph control allows for the customization of the displayed information; including multiple curve overlays, defining curve types, markers, labels, etc.

Bi-directional optical signal analysis

For optical signals, both the input (ingress) and output (egress) signals can be viewed for any optical probe. It is also possible to change the format (on a per probe basis) of the optical data format including viewing optical field information in complex, magnitude/phase, power or chirp representations.

Data export

The data captured from any probe can be exported for visualization or manipulation by 3rd party tools such as calculation spreadsheets.

Presentation and analysis tools

All 2D Graphs can be modified (multiple curve overlay, curve types, markers, axes labels, legend, etc.) to assist with report preparation. Graphs can be saved to clipboard or exported in BMP or EMF formats.

For further information on the Waveform Viewer please refer to the “OptiSPICE Waveform Viewer” feature description under *Help/Help Docs/OptiSPICE Waveform Viewer*.

OptiSPICE Samples Installer

Within Windows Explorer, under All Programs/Optiwave Software/OptiSPICE 4, a new “Install OptiSPICE Samples” has been introduced. Simply run the application and a full copy of the OptiSPICE 4.0 samples directory will be created at the requested file path (see Figure 3B). The procedure can be repeated as many times as required!

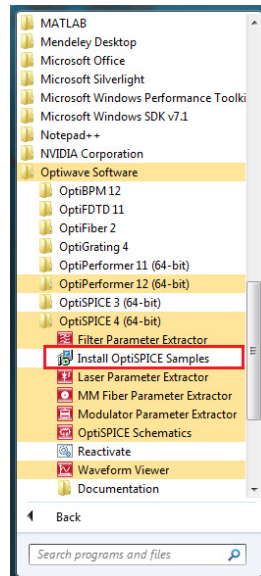


Fig 3A: New “Install OptiSPICE samples” feature.

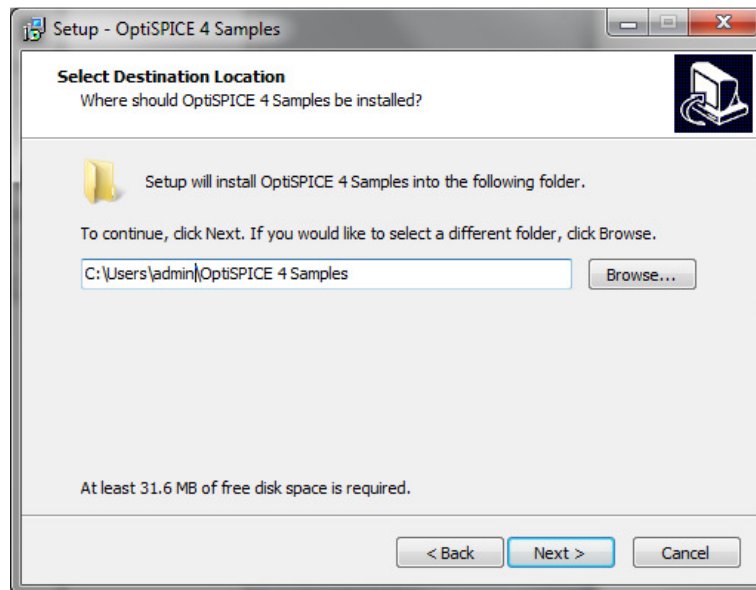


Fig 3B: Definition of installation directory for “Install OptiSPICE samples” feature

Other OptiSPICE improvements

Previous issues with AC Analysis for optical domain signals have been resolved. OptiSPICE now supports, on top of DC, operating point and transient analysis; AC analysis for all optical and opto-electronic circuits.

Previous versions of OptiSPICE were dependent on installing OptiSystem to allow for the viewing of post-simulation data. This requirement no longer applies to OptiSPICE 4.0. Users need only install OptiSPICE for full post-simulation analysis and if desired (if OptiSystem is installed on the same computer) seamlessly access OptiSystem visualizers through the new “Launch OptiSystem” button in the OptiSPICE simulation results window (see Figure 4).

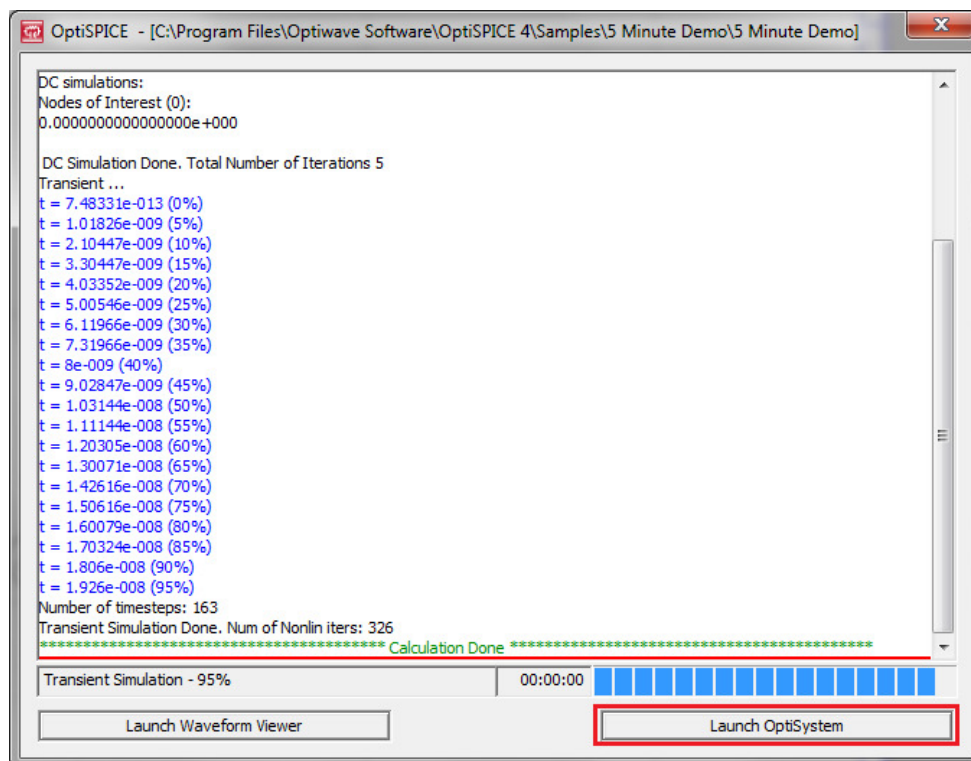


Fig 4: New “Launch OptiSystem” feature.