

# HyperLynx Analog

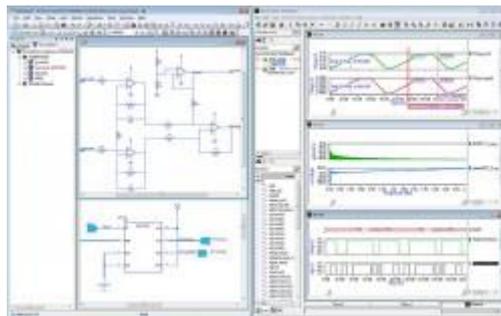
Full-board functional simulation with SPICE, VHDL, Verilog and other industry-standard languages

## Overview

HyperLynx® Analog delivers truly scalable analog circuit simulation, allowing PCB designers to precisely control simulation parameters and environment.

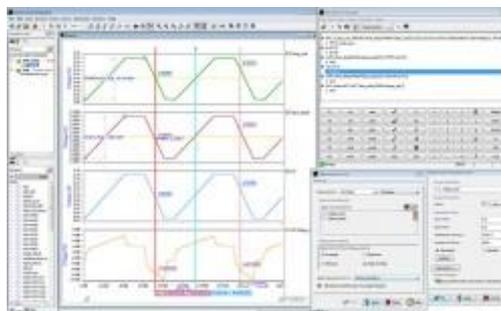
It extends the capabilities of DxDesigner to include mixed-signal and analog circuit design simulation to significantly cycle time and eliminate design transcription errors.

By leveraging functional simulation early in the design process, design teams can explore alternate circuit technologies and control component cost and manufacturing yields with much greater efficiency than with common prototype-based processes.



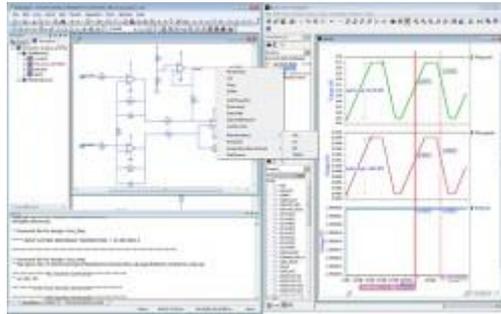
## Perform full board functional simulation using industry standard languages

Provides industry standard mixed-signal simulation using SPICE, VHDL, Verilog, Verilog-A, VHDL-AMS and Verilog-AMS languages



## Perform full board functional simulation using industry standard languages

Provides industry standard mixed-signal simulation using SPICE, VHDL, Verilog, Verilog-A, VHDL-AMS and Verilog-AMS languages



### One common schematic for simulation and PCB design

Single schematic helps eliminate design transcription errors and expedite time to market for your products

## Expand analysis capabilities to eliminate waste

HyperLynx Analog helps you perform mixed-signal and analog circuit design simulation for greater analysis capabilities, shortening the development cycle and reducing waste and manufacturing costs.

- **Easy-to-use environment**
  - Guided simulation – The flow-centric toolbar leads the engineer through the simulation process.
  - Project browser – HyperLynx Analog adds a test bench browser that supports multiple test benches per design block; the user may create multiple design blocks in DxD Designer, and functionally verify them under various conditions with all available analyses
- **Sources definition** – Define simulation sources outside of the schematic maintaining the schematic content integrity for the PCB layout process and set up different source configurations for multiple test benches and simulation runs
- **Centralized simulation control** - HyperLynx Analog provides a single dialog to set up DC, AC, transient analyses and then extend the basic simulations to parametric sweeps, Monte Carlo, noise and sensitivity analyses, without learning complex commands
- **Rapid import of web-based simulation models** - HyperLynx Analog supports a rich feature set to allow design engineers to access the latest components available through the internet
- **Model editing** - To support expert users, HyperLynx Analog provides syntax-sensitive text editors allowing the engineer to easily create or edit simulation models

- **Complete modeling solution**
  - Provides quality models in an intelligently partitioned library for design acceleration
  - Maintains SPICE standards for easy imports from component vendor's web-based models for quick design inclusion
  - Supplies a native PSpice model conversion utility
- **Scalable simulation options**
  - Provides a flexible range of simulation algorithms from an embedded SPICE simulator for quick analyses to the IC market-proven Eldo SPICE-based simulation engine for extreme capacity and blistering-fast performance:
    - Includes SPICE support with improved convergence and performance compared to other commercial SPICE simulators
    - Eldo simulation engine option adds HSPICE compatibility and IBIS support and superior convergence and performance
    - Extends into true mixed-signal domain board systems design; Questa ADMS single kernel, co-simulation engine provides industry standard language based mixed-signal simulation using the SPICE, VHDL, Verilog, Verilog-A, VHDL-AMS and Verilog-AMS languages
    - Scales to your simulation needs, using SPICE-standard spanning IBIS, VHDL, Verilog, Verilog-A, VHDL-AMS and Verilog-AMS
    - Collaborates with HyperLynx Signal Integrity to extract PCB trace parasitic models from the physical layout and incorporate them into the functional simulation when using either the Eldo or Questa ADMS simulation kernel upgrades
- **Improves design quality and manufacturability**
  - Provides the standard DC, transient and frequency analyses
  - Allows sweeping component values as well as sources facilitates fast optimization of component values
  - Statistical Monte Carlo and worst-case analyses enable the engineer to identify components that most affect circuit performance and predict likely design yields by considering tolerance variations across components
- **Delivers advanced results analysis**
  - Includes waveform viewer for interactive measurement of design analysis and verification that works on a unified waveform database with digital and analog simulation data saved in a single compact and efficient representation
    - Allows side-by-side comparison of multiple simulation runs while also providing easy navigation of all of the nodes in the design using either a hierarchical or flat representation
    - Includes measurements between transition points through the use of multiple cursors and interactive event search, creation of special diagrams and charts (eye diagram, smith chart, XY plot,...), and waveform post-processing using

built-in (complex, logic, mathematical, trigonometric, etc) or user-defined functions