

## SPISimPro Design Concept:

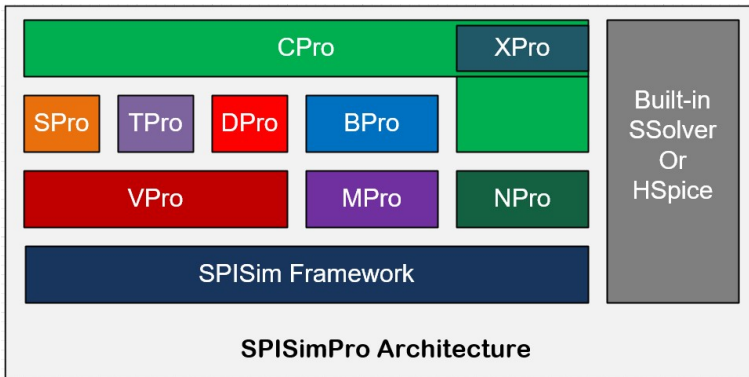
To perform system electrical analysis, one needs to create and/or prepare device models in the forms of IBIS, AMI, S-parameters and transmission lines. These models are then assembled as schematic either for pre-layout analysis or extracted from post-layout design by nets. Simulation or statistical analysis are then performed and results are post-processed and reported. Besides, systematic approaches such as design-of-experiment (DOE) flow are usually required in order to create a response surface model (RSM) or an artificial neural net (ANN) from sampling points in solution space for design optimization. For DDR, JEDEC parameters may need to be computed to check for spec. compliance.

SPISim has modules covering full spectrum of signal, power integrity analysis steps mentioned above. From pre-layout, post-layout to lab data processing, add-ons and capabilities are ready to minimize your effort and maximize the efficiencies. SPISimPro integrate all modules and run seamlessly on a unified framework across different platforms. We can also further customize the tool to meet your team's design challenge.

## APPLICATION SCOPES:

SPISimPro enables each step of signal integrity analysis from modeling, pre-layout, simulation, post-layout, optimization to automatic data processing and reporting:

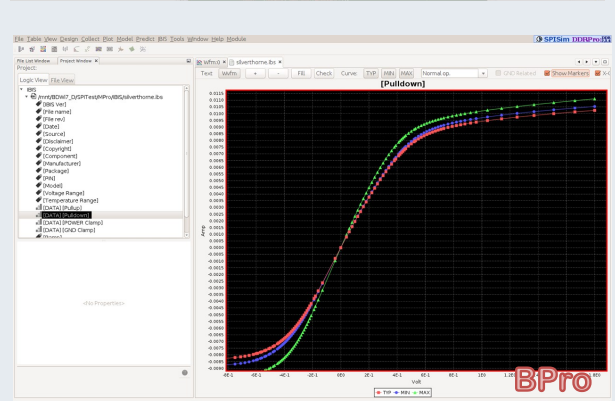
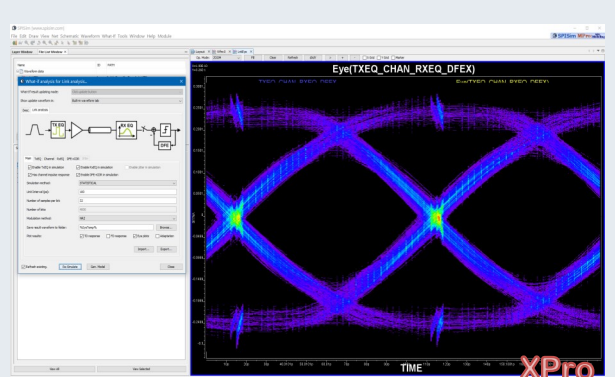
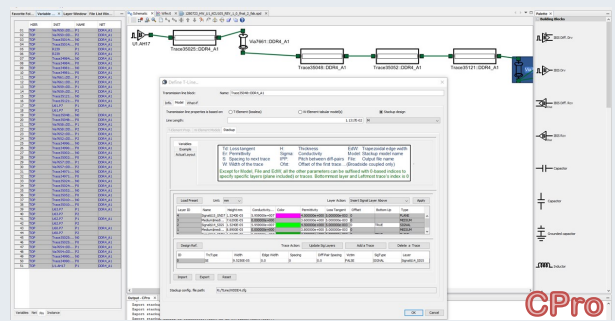
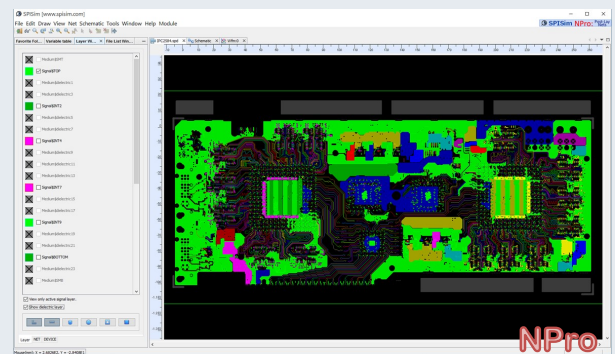
- Extract nets from post-layout to view/simulate interactively or doing what-if;
- Free-form schematic channel building to simulate or doing what-if analysis;
- Powerful TD/FD simulation waveform viewing, measurement and analysis;
- Comprehensive S-parameter analysis, modeling, viewing and report generation;
- Layer stackup modeling and transmission line analysis with user's or HSpice 2D solver;
- IBIS V3.2 ~ V5.1 modeling generation, analysis and reporting w/ built-in checker. Supports both single-ended and differential models of all modeling types. Convert from spice to IBIS and IBIS to spice as well.
- DOE/RSM/ANN etc analysis, modeling, visualization and optimization;
- DDR simulation waveform processing to extract JEDEC Spec. parameters;
- What-if analysis for IBIS, via, T-Line, package and full link analysis. Generate model including spec. AMI model instantly.
- Support HSpice simulator, 2D field solver, and schematic S-parameter extraction.



## SPISimPro includes the following modules:

- **NPro:** Post-layout data review and net extraction from MCM/BRD/ODB++ design. Extract net(s) to CPro's channel builder.
- **CPro:** Schematic channel builder. Build free-form channel from scratch, template or edit channel extracted from post-layout nets. Perform TD/FD interactive or what-if analysis.
- **VPro:** FD/TD waveform viewer w/ signal generator. View/measure/calculate simulation or measurement data. Support 10+ formats and scripting. Spectrum analysis with windowing.
- **SPro:** S-Parameter focused modeling with dedicated S-parameter viewer with 20+ analysis capabilities plus automatic report generation. Perform TDR/TDT/PDA in single click.
- **TPro:** Transmission line focused modeling features with dedicated T-Line model viewer and stackup planner with built-in database of > 100K pre-solved structures for instant what-if.
- **BPro:** IBIS focused modeling with IBIS model inspector with model waveform editor. Step-by-step to create models from Spice to IBIS or vice versa. Supports all model types including differential and series elements. View MOSFET in 3D.
- **DPro:** DDR simulation data processing. Post-process DDR simulation results with auto derating to extract JEDEC spec. based measurements.
- **MPro:** General modeling and optimization. Perform DOE/RSM/ANN modeling and more. Optimize with linear programming or genetic algorithm for best design parameters.
- **XPro:** What-if analysis for IBIS, via, T-Line, package decoupling and link analysis. See changes instantly and generated associated models including IBIS and spec. AMI models.
- **SSolver:** built-in spice simulator, s-parameter extraction via AC analysis. Support XSpice for extension.

## Screenshots:



and more...

<http://www.spisim.com/products/SPISimPro/>

Info: [info@spisim.com](mailto:info@spisim.com)

Sales: [sales@spisim.com](mailto:sales@spisim.com)

All Rights Reserved. Copyright 2009-2019, SPISim LLC, USA

