

Pulsonix Spice Simulator

Advanced A/D Mixed Mode Simulator

Pulsonix Spice is a low cost, advanced mixed-mode circuit simulation package delivering exceptional performance in terms of convergence reliability and speed. This exciting product is fully integrated into the Pulsonix Schematics environment.

Superior Convergence and Speed

Pulsonix Spice is based on a substantially enhanced version of SPICE 3 and XSPICE. The underlying algorithms have been reworked to provide new analysis modes to improve convergence and speed. In the case of convergence, in trials Pulsonix Spice out performed three well known Spice-based products including the market leader. These major advances have been achieved with proprietary enhancements to the transient analysis algorithms and the development of automatic pseudo transient analysis.

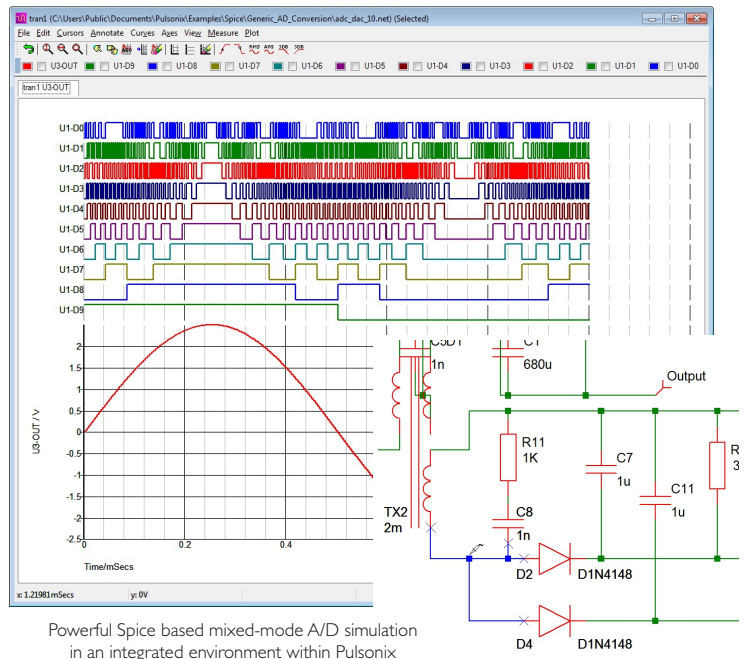
Using a set of 57 standard industry circuits, Pulsonix Spice outperformed other Spice simulators. Most of these simulators are even several times the price of Pulsonix Spice despite not being high performers in terms of successful simulation convergence and completion during these benchmark trials.

Advanced Waveform Analysis

A comprehensive selection of analysis functions may be applied to data plotted. At the click of a mouse the RMS, rise and fall time, -3dB point or many other functions can be calculated and displayed alongside the graph legend. Many of the available functions can be optionally applied to a portion of a waveform defined by cursor positions.

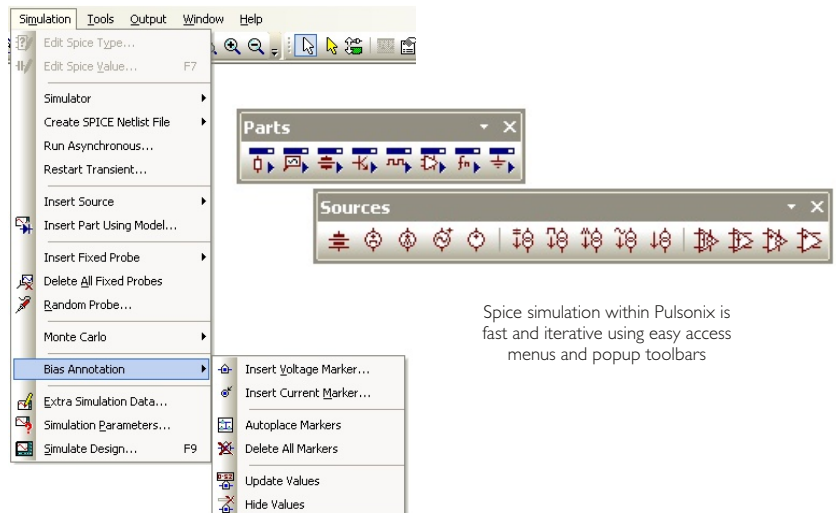
Pulsonix has extended sweep modes. Standard SPICE has just one sweep mode for each of the analyses AC, DC and Noise whereas Pulsonix Spice has six. For example, in AC analysis, Pulsonix Spice allows you to sweep a device value such as a voltage source at a fixed frequency so you can plot the gain of an amplifier Vs bias.

The Noise mode performs a small signal noise analysis of the circuit linearised about its operating point, calculates total noise at a nominated output and the contributions from every noisy device. This operates in any of the six modes available for AC analysis.



Random Probing

Pulsonix Spice includes post simulation random-probing. Following successful simulation, graphs of circuit voltages, currents and device powers can be created simply by clicking on a point on the schematic with the probe. This avoids the need to re-simulate the circuit each time an additional measurement is required. Alternatively, probes can be permanently fixed to a schematic so that a graph is created and incrementally updated during the simulation.



Spice simulation within Pulsonix is fast and iterative using easy access menus and popup toolbars

Pulsonix Spice Analysis Modes

Operating Point	Finds steady state bias point and produces report of circuit voltages, currents and device operating parameters Analysis over time. Provides voltage, current and power at all nodes or devices
Transient	Performs a small signal analysis of the circuit linearised about its operating point
AC	Operates in one of six modes: <ul style="list-style-type: none"> ■ Frequency sweep. (As standard SPICE) ■ Device sweep. (E.g. resistor or capacitor value) ■ Model parameter sweep ■ Global parameter sweep ■ Temperature sweep ■ Single step Monte Carlo sweep (repeats analysis while applying component tolerances)
DC	Repeats DC solution using any of the sweep modes, except frequency, described for AC analysis
Noise	Performs a small signal noise analysis of the circuit linearised about its operating point. Calculates total noise at a nominated output and the contributions from every noisy device. Operates in any of the six modes described for AC analysis
Transfer Function	Similar to AC but calculates response to a single output from all sources. Operates in any of the six modes described for AC analysis
Real Time Noise	An extension of transient analysis, applies noise generators to all noisy devices with a magnitude calculated using the same equations as for small signal noise. Allow noise plotting in real time
Sensitivity	Calculates sensitivity of circuit components and model parameters to a specified circuit parameter
Pole-Zero	Finds the AC transfer function of a circuit in terms of its pole and zero locations

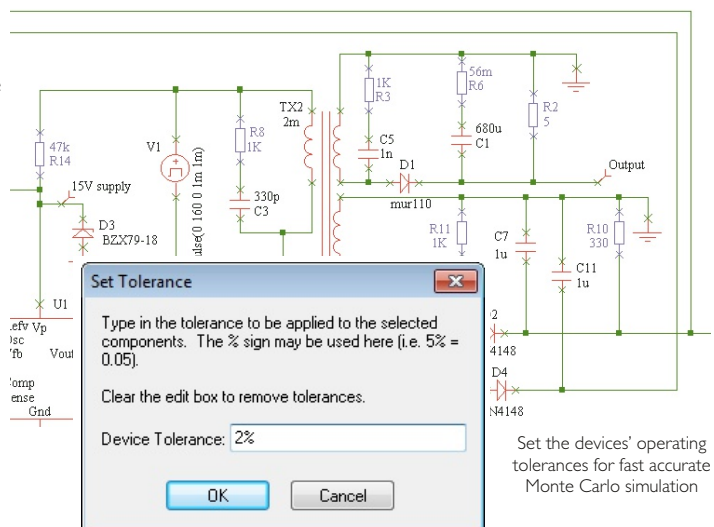
Monte-Carlo Analysis

Pulsonix Spice has one of the fast Monte Carlo analysis modes available for PC based products. Unlike other simulators, the Pulsonix Spice Monte Carlo feature is built into the simulator core. This shortens the loop and avoids unnecessary repetition of some of the stages in the simulation such as the netlist read in and set-up phases. In addition, the DC bias point at each step is speeded up by seeding the solution with the result of the previous one.

Monte Carlo analysis enables you to simulate real life; where components operate within tolerance bandwidths. The percentage tolerance is entered and the circuit is simulated repeatedly with the component's operating tolerances automatically varied on each simulation run. The resultant compound waveforms enable you to determine if the circuit still performs within design specifications when tolerances are varied.

Integrated Library

Pulsonix shares the same set of libraries for Simulation, Schematic capture and PCB layout, there are no additional or separate libraries required when moving between each process. Spice device models are quickly referenced to any required Part in your library ensuring standardisation and accuracy across your systems.



Pulsonix Spice Feature Summary:

- Integrated into the Pulsonix Schematic capture design environment
- Easy to use and understand dialog driven user interface
- 30,000 Spice model definitions and 6,500 actual models supplied
- Drag and drop model addition to the Pulsonix library
- Comprehensive waveform analysis
- User definable scripting language in simulator
- True mixed-mode simulation
- Safe Operating Area (SOA) limits for use within DC or Transient
- HSpice model file compatibility
- Compatibility with most SPICE models available on the internet