

Pulsonix Spice Simulator

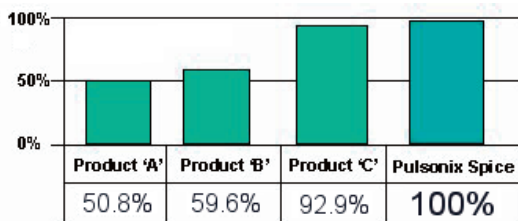


Advanced A/D Mixed Mode Simulator

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 recent trials Pulsonix Spice out performed 3 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.



The above graph shows percentage completion of a set of 57, industry recognised, standard benchmark circuits. Products 'A' and 'B' are Spice based simulators from well known suppliers and product 'C' is widely regarded as the industry standard product for Spice simulation, and is marketed at several times the price of Pulsonix Spice. As can be seen, Pulsonix Spice out-performed all these products in terms of successful simulation convergence during these benchmark trials.

Advanced Waveform Analysis

A comprehensive selection of analysis functions may be applied to plotted data. 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.

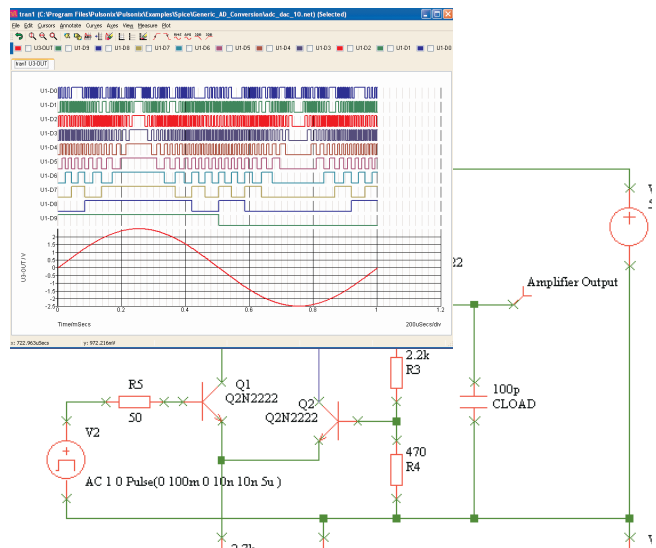
Many of the available functions can be optionally applied to a portion of a waveform defined by cursor positions.

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. Operates in any of the six modes described for AC analysis

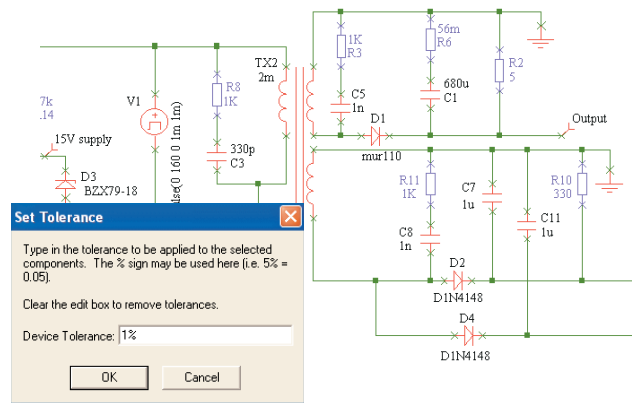
Random Probing

Pulsonix Spice includes post simulation random-probing. Following simulation, graphs of circuit voltages, currents and device powers can be created simply by clicking on a point on the schematic. 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.

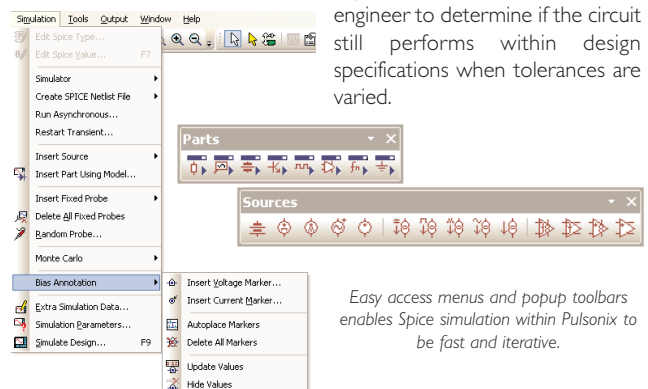


Monte-Carlo Analysis

Pulsonix Spice has possibly the fastest Monte Carlo analysis of any PC based product. 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. Further, 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 the engineer 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 the engineer to determine if the circuit still performs within design specifications when tolerances are varied.



Easy access menus and popup toolbars enables Spice simulation within Pulsonix to be fast and iterative.

Pulsonix Spice Simulator

The 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

Pulsonix Spice at a glance

- Integrated into the Pulsonix Schematic capture design environment
- Dialog driven user interface
- Convergence performance in benchmark trials exceeds results from industry leaders
- True mixed-mode simulation: closely coupled direct matrix (SPICE 3) analogue and event driven digital simulator
- Post simulation random probing
- Monte-Carlo and Multi-Step analysis
- Noise analysis
- Transient restart
- Non-linear magnetics with support for air gaps
- Automatic pseudo transient analysis algorithm for operating point solution
- Variable step GMIN and source stepping (the standard SPICE3 variants use a fixed step)
- Bias annotation markers with dynamic updating
- 30,000 Spice model library definitions and 6,500 actual models supplied
- Drag and drop model addition to the library
- Pulsonix Spice is compatible with 99% of available SPICE models
- Import standard SPICE models from outside suppliers - many of which can be downloaded from the Internet
- HSpice model file compatibility
- Analyses: operating point, DC sweep, transient, AC small signal, transfer function, sensitivity, pole-zero
- Comprehensive waveform analysis
- User definable scripting language in simulator
- Comprehensive printed user reference guide

Pulsonix Oak Lane, Bredon, Tewkesbury, Glos, GL20 7LR, UK
Tel: +44 (0) 1684 773881 Fax: +44 (0)1684 773664
Email: sales@pulsonix.com Web: www.pulsonix.com