



## 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 new 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 convergence during these benchmark trials.

### **Advanced Waveform Analysis**

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.

Real time noise analysis. An extension of transient analysis that enables real noise sources for all noisy devices using the same equations as small signal noise analysis. This unique feature makes it possible to analyse noise in oscillators and sampled data systems.

## **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.



Random probing avoids the need for resimulation

#### 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, components operate within tolerance bandwidths. where 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.

# www.pulsonix.com

## 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: 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
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 Pulsonix Schematics 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 analysis
- Noise analysis
- 25,000 Spice model library, plus can import standard SPICE models from outside suppliers many of which can be downloaded from the Internet
- Analyses: operating point, DC sweep, transient, AC small signal, transfer function, sensitivity, pole-zero
- Comprehensive waveform analysis