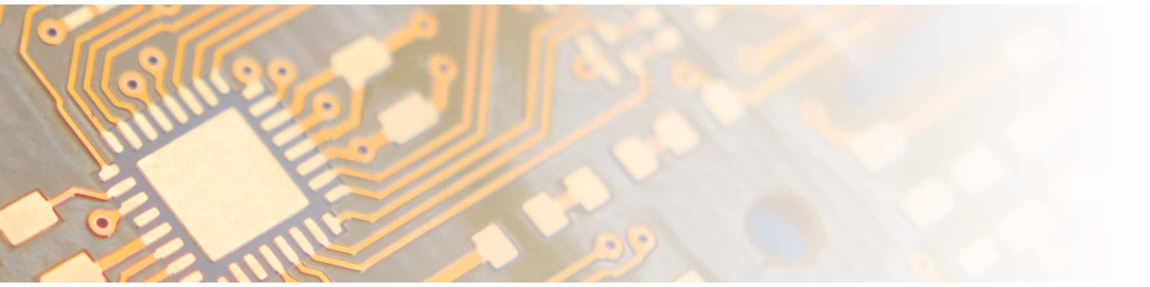
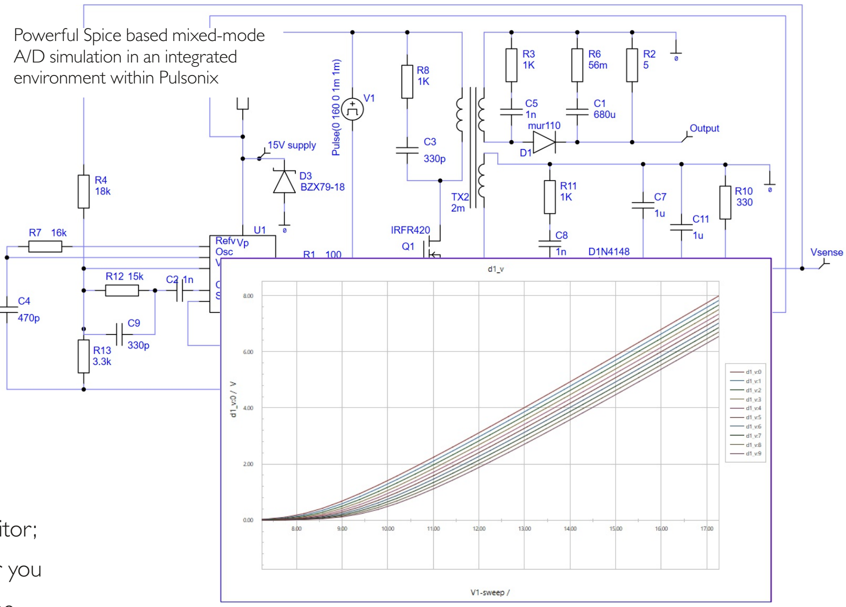


PulsonixSim Spice Simulator

Advanced Spice A/D Mixed Mode Simulator

PulsonixSim is a low cost, Spice-based mixed-mode circuit simulation package for analysing and visualising the characteristics of your Schematic design. Presented in a user-friendly dialog-driven interface, it delivers outstanding performance in terms of convergence, reliability and stability using the industry standard ngspice engine.

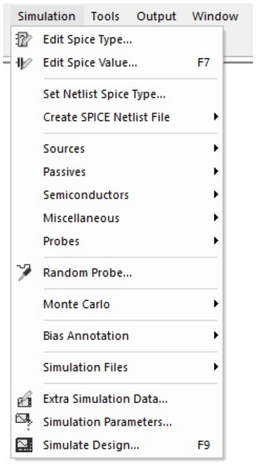
The ngspice core code used is based on three open source software packages: Spice3f5, Cider1b1 and Xspice. It is the open source successor of these venerable packages. Many, many modifications, bug fixes and improvements have been added to the code, yielding a stable and reliable simulator.



Powerful Spice based mixed-mode A/D simulation in an integrated environment within Pulsonix

Integrated Dialog Driven Interface

The simulator is integrated and driven from Pulsonix Schematic editor; no netlists to save and load into the engine, this is all performed for you with the results displayed in Pulsonix using the inserted graphs in the design and full graphs.



Spice simulation within Pulsonix is fast and iterative using the easy access menu and simulation toolbar

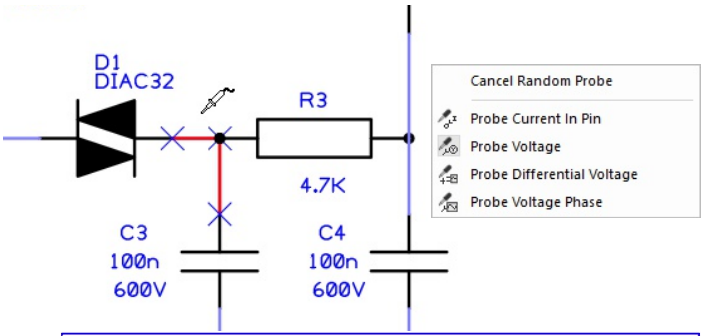
Integrated Libraries

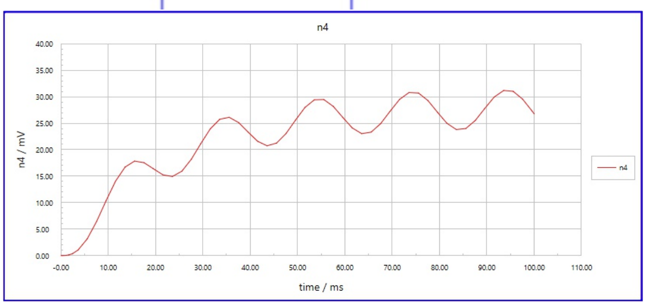
Pulsonix shares the same set of libraries for Simulation, Schematic capture and PCB layout; there are no additional or separate libraries required when transitioning between each stage of the design process. Spice device models are quickly referenced to any required Part in your library ensuring standardisation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

Schematic designs can be simulated using the same libraries that will be seamlessly passed to the PCB design. Spice data is simply attached to the Part definition in the library with the manufacturers Spice models also saved and stored within the Pulsonix library structure.

Random Probing

PulsonixSim includes post simulation random-probing. Following successful simulation, graph windows of circuit voltages and currents 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.





Parts and Stimuli

PulsonixSim is supplied with over 150 primitives provided containing Stimuli, probes, sensors, primitives and devices. It also includes a selection of analog and digital devices, from passive components to transistors, diodes, and analog ICs, the library provides real device models. For digital designs, the library includes flip-flops, gates and more comprehensive ICs. This makes PulsonixSim an ideal tool for mixed-mode simulation.

Spice Models

PulsonixSim is supplied with around 9000 manufacturers' Spice models and sub-circuits. All PulsonixSim Parts supplied have real models already defined for them. The simulator is compatible with most Spice models from external suppliers available many of which can be downloaded from the Internet. External sourced models can be easily added to existing Pulsonix Parts.

Model Compatibility Support

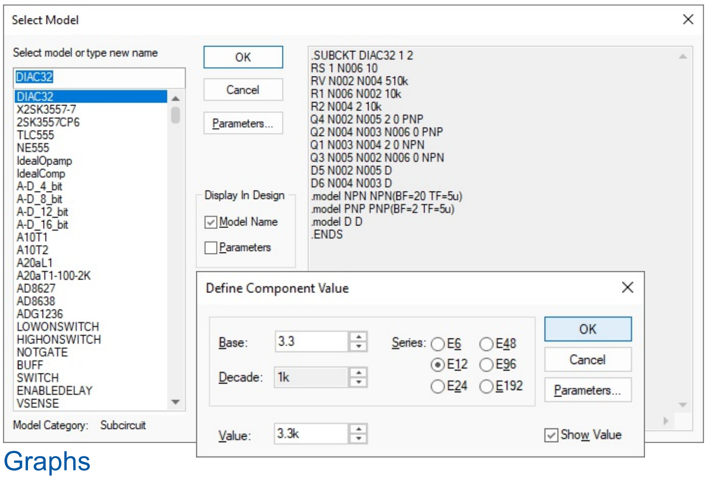
PulsonixSim is compatible with most PSpice and LTSpice device model parameters and netlists for simulating discrete circuits. PulsonixSim will also read HSPICE device libraries from semiconductor foundry PDKs for simulating integrated circuits.

Parts Browser For Fast Searching

Searching and navigating the libraries is made easy with the graphical Parts Browser bar. Parts can be searched by Part number or by Spice Value, other categories can be used based on your choice. The standard categories have been organised for logical functional groups.

Search by Model

A special mode in PulsonixSim allows you to search Spice models based on their Spice type. This enables you to create one master Part and to use it for multiple Spice instances. For example, passives such as Resistors, Diodes, Zener Diodes etc. can be generically defined which means less library clutter.



Graphs

Graphs can be in-design windows or full window depending on how much detail you need to view. Full windows can be undocked from the main application and pinned to another screen when detailed inspection of simulation results are required. In-design graph can be positioned and resized along with customisation of detail such as colour and graph idents. These can also be combined to display two or more sets of overlaid results.

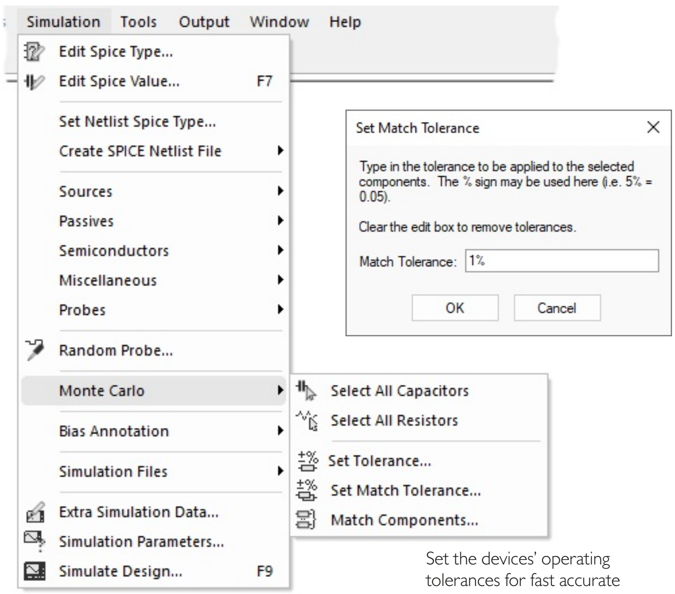
Probes can be permanently fixed to a schematic so that a full window graph is created and incrementally updated during the simulation.

Circuits can be simulated without probes, in which case all voltages and currents are determined. Graphs can be added at specific points and the results used.

Graph probes attached to individual nets can be moved at any time and attached to another net to display the results at that point. They can also be attached to a component to display the results for all pins on that device.

All graph results can be exported to an SVG file format for use in your documentation.

Once a design has been simulated, the results and graphs are also saved when the design is saved, meaning the circuit doesn't need re-simulating when you open it again or open it on another machine.



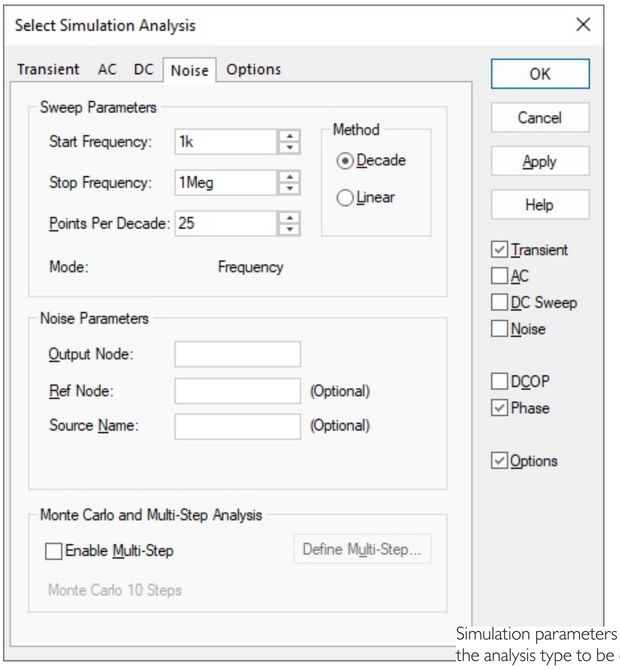
Set the devices' operating tolerances for fast accurate Monte Carlo simulation

Monte-Carlo Analysis

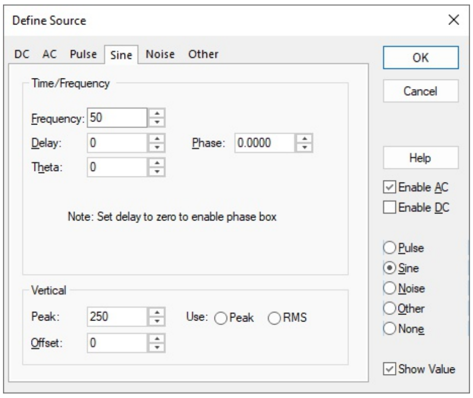
Monte Carlo analysis enables you to simulate real life; where components operate within tolerance bandwidths. The percentage tolerance is entered and the circuit is simulated repeatedly with the component's operating tolerances automatically varied on each simulation run. The resultant compound waveforms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

PulsonixSim 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 and current at all nodes or devices
Transient	Performs a small signal analysis of the circuit linearised about its operating point
AC	Operates in one of these modes: ■ Global parameter sweep ■ Device sweep ■ Model parameter sweep. (E.g. resistor or capacitor value) ■ TemperatureSweep ■ Frequency sweep ■ 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 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 modes described for AC analysis
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	Calculate the DC operating-point sensitivity or the AC small-signal sensitivity of an output variable with respect to all circuit variables, including model parameters
Pole-Zero	Finds the small-signal AC transfer function of a circuit for its pole and zero locations



Simulation parameters enable the analysis type to be defined



Each source can be defined providing accurate stimuli for the circuit

Viewing Simulation Files

Using the Simulation menu, the Spice netlist file generated, the log file of activity and the output results file can all be easily viewed in your text editor. These files often give important clues if your circuit fails to simulate. It also reports if Spice models are missing from a Part definition or even from the model search path.

Pulsonix Spice Feature Summary:

- Integrated into the Pulsonix Schematic capture design environment
- Easy to use and understand dialog driven user interface
- True mixed-mode simulation
- Over 150 ready-to-use Spice Parts and primitives
- 9,000 Spice model definitions and sub-circuits supplied
- In-design graphs and full graph windows
- Results saved with design for instant reuse
- Post simulation random probing
- Monte-Carlo and Multi-Step analysis
- Noise analysis
- Phase for use within AC analysis
- Bias annotation markers with dynamic updating
- PSpice, LTSpice and HSPICE model compatibility
- Compatibility with most SPICE models available on the internet

Pulsonix

20 Miller Court, Severn Drive, Tewkesbury, Glos, GL20 8DN, UK

Tel: +44 (0) 1684 296 551 Email: sales@pulsonix.com Web: www.pulsonix.com