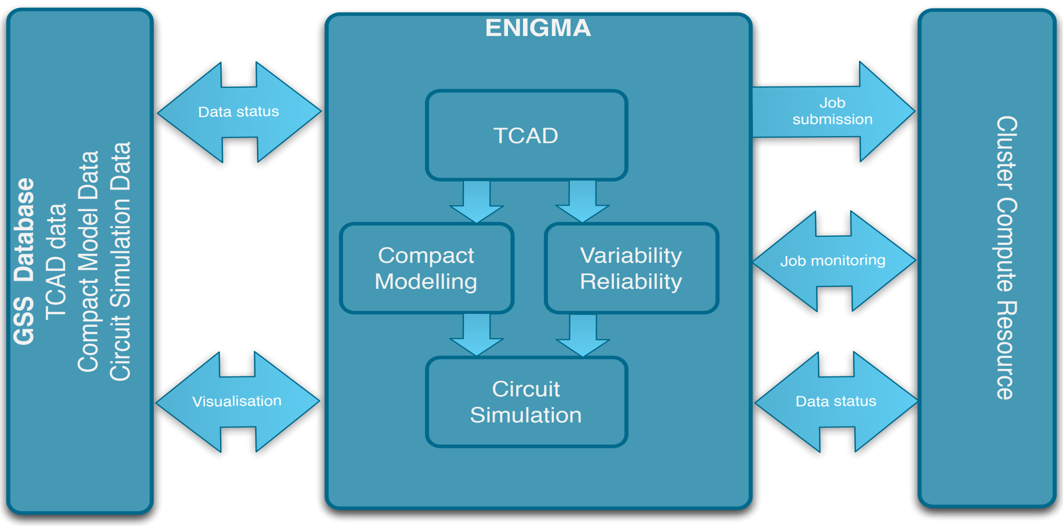
Enigma

Workflow and Automation Framework

Product Description

Enigma is the GSS automation and productivity framework, which has been designed from the ground-up to deliver a powerful and flexible environment for rapid design-technology co-optimisation (DTCO). It offers the capability to quickly evaluate technology changes and their impact on performance and yield. Furthermore, it offers the capability to automatically generate PDK-quality compact models based on predictive TCAD simulations.

Enigma enables users to easily and transparently submit, manage and monitor large-scale simulations (tens to hundreds of thousands) on computational clusters (SGE, LSF). This makes process exploration and variability studies straightforward to launch and process. Analysis and plotting modules are included that provide tools to automatically generate report and publication-ready graphs and images.



In addition to this, Enigma also provides an optimisation module that enables fully automated calibration of Garand to measurement or Monte-Carlo simulations.

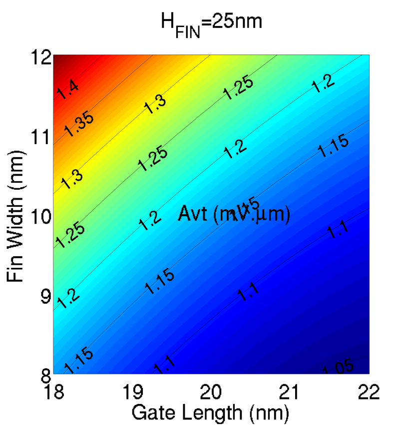
In Enigma, simulation flows are encapsulated as Python scripts, which ensures full reproducibility and all simulation data are captured in a fully integrated database. The simulation flows and resulting data are automatically managed by Enigma, ensuring data consistency and reliability. This allows complete simulation flows from transistor level simulations through compact model extraction and circuit simulation.

For further information please contact: *sales@GoldStandardSimulations.com*

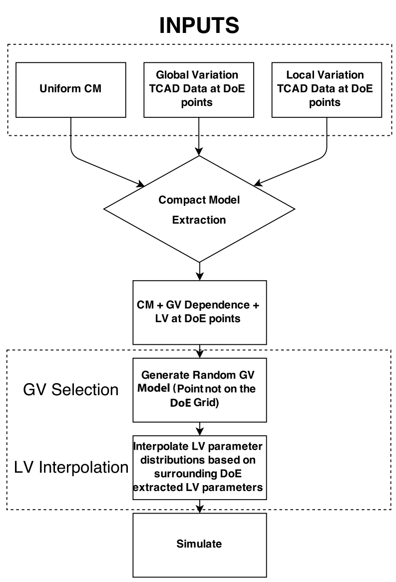
Enigma is the optimal tool flow solution for TCAD/DTCO/PDK development by automating the steps of the TCAD to circuit flow.

Through its simple and intuitive scripting interface, Enigma enables the generation of all the data required for transistor design optimisation.

For example, the graph below shows the Avt factor dependence on Fin design parameters Fin length, Fin width and Fin height. This data was generated with a single loop command in Enigma.

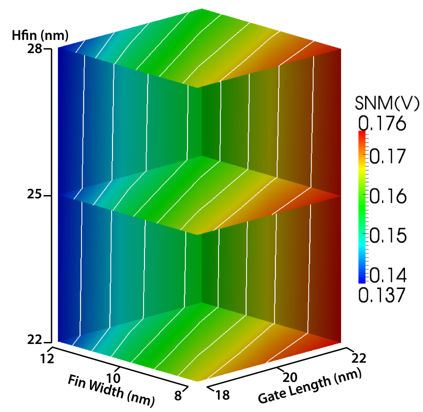


Enigma simplifies the generation of the massive amount of data required for accurate compact model extraction, as well as ensuring the correct data is used in the extraction strategy.



The extracted models can then be used to generate extremely accurate compact models including global and local variability and the correlation between these (which is extremely important in FinFET processes). This process is shown in the image above.

In the case of DTCO, the transistor redesign impact on critical circuit performance can be evaluated using any user defined testbench circuits. This is critical, especially for highly sensitive circuits like SRAM bitcells, where minimal area and sensitivity to local variability can significantly impact power, performance and yield.



Rapid iteration is enabled by Enigma's advanced dependency resolution techniques, which handle the interactions between the different steps of the TCAD/DTCO/PDK development flow and will only re-run the necessary steps that impact the final output.

Enigma works within most common cluster environments, allowing for smart resource allocation and monitoring/reporting of job status, **allowing the engineer to focus on design and analysis**.

The focus on productivity, automation and simplification makes Enigma a must have tool for anyone looking to rapidly co-optimise device and circuit performance, evaluate future technologies, or seek to gain market differentiation through improved early PDK accuracy.

Enigma also offers the ability to support 3rd party applications through its own scripting interface, or through custom modules developed and supported by GSS.

Features

- Automated flow for TCAD to circuit simulation
- Built in optimisation for device calibration
- Sophisticated database for easy data management and tracking
- Status monitoring and quick reporting
- Simple scriptable Python interface for easy automation and repeatability of experiments
- Multiple users working in the same environment ensuring consistency within a development team
- Compatible with all major cluster management tools (LSF, SGE)
- Dependency checking to ensure minimal CPU wastage
- Automatic figure plotting and data comparison
- Optional 3rd party tool support

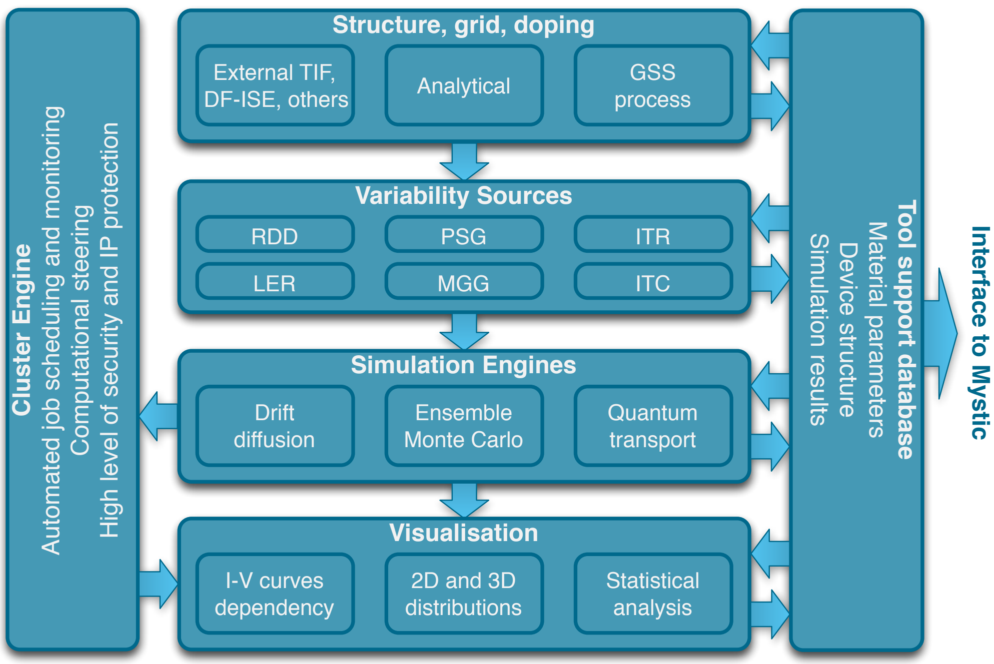
For further information please contact: *sales@GoldStandardSimulations.com*

Garand

Advanced 'Atomistic' TCAD Simulator

Product Description

GARAND is the GSS 'atomistic' TCAD device simulator. It is perhaps the most advanced TCAD simulator available, allowing predictive simulation of present and future CMOS transistors including bulk and FDSOI MOSFETs, FinFETs and Nanowire transistors. GARAND has 3D Drift Diffusion (DD), Ensemble Monte Carlo (EMC) and Non-Equilibrium Green's Function (NEGF) modules.

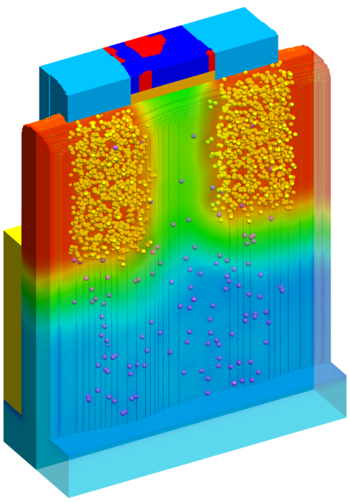


Overview of the Garand Simulation Tool

The DD, EMC and NEGF modules share identical simulation domains and allow physical simulation of all relevant variability sources including Random Discrete Dopants (RDD), Line Edge Roughness (LER), Poly-Silicon and Metal Gate Granularity (PSG and MGG). A Kinetic Monte Carlo engine allows simulation of time dependent variability associated with trapping/de-trapping of discrete charges on defect states.

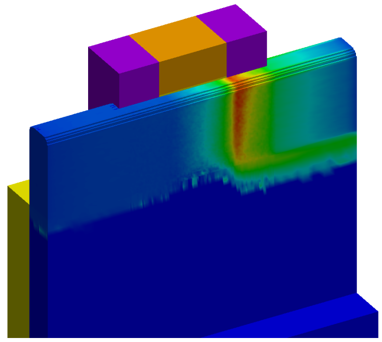
Accurate Density Gradient (DG) quantum corrections guarantee the resolution of each individual Coulomb charge in the simulations and account for quantum confinement effects that dominate the latest 3D device architectures such as FinFETs and nanowire MOSFETs. 2D Poisson-Schödinger quantum corrections are now also available. The full 3D quantum corrections are also fully maintained within the Ensemble Monte Carlo simulations.

The **Drift-Diffusion** module of Garand has been designed from the ground up for the accurate simulation and analysis of intrinsic statistical variation in advanced CMOS transistors introduced by the discreteness of charge and granularity of matter. All of the important sources of statistical variability can be introduced in GARAND through a single command line, and can be simulated individually and in arbitrary combinations. Statistical reliability effects associated with capturing of discrete charges on defect states in the oxide and the generation of new defect states can be seamlessly simulated with GARAND.

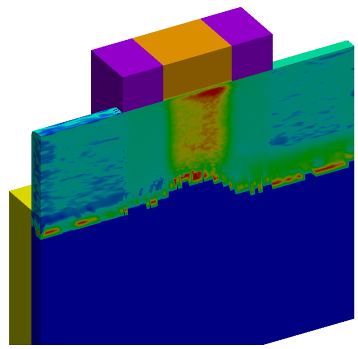


Accurate physical simulation of statistical variability in 10nm CMOS FinFET including RDD, LER and MGG

The **Ensemble Monte Carlo** module of GARAND is the only fully self-consistent 3D quantum-corrected full-band Ensemble Monte Carlo transport engine available commercially. It is designed from the ground up and developed over a decade to provide accurate device performance prediction in modern CMOS transistors governed by non-equilibrium, quasi-ballistic transport. It is indispensible when analyzing and screening future technology options including new transistor architectures and new channel materials. Fermi-Dirac statistics and Pauli's exclusion principle enable the simulation of degenerate materials. Full self-consistency and direct carrier-carrier and carrier-dopant interactions ensure accurate results, capturing coupled carrier interactions and extending the physical model beyond any other commercially available simulator.

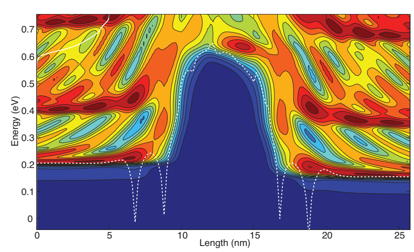


Electron energy distribution in a 10nm CMOS FinFET from EMC simulation



Electron velocity distribution in a 10nm CMOS FinFET from EMC simulation

The ballistic **NEGF** module has full 3D capabilities based on a recursive algorithm and a fast coupled-mode-space option. A Coupled Mode Space NEGF module based on 2D cross-sectional Poisson/Schrödinger solution includes phonon scattering. It is also a basis for the current development of a 1D multi-subband MC simulator for FinFETs and nanowire transistors.



Density of States distribution from NEGF simulation of a nanowire transistor

Tight integration with the GSS Job Management tools allows the statistical simulation of large ensembles of transistor characteristics on an unprecedented scale. Such large-scale simulations are vital to accurately capture the extremes of device performance that can facilitate high-sigma analysis of circuits and systems.

Features

- Transfer of device structures from third party process simulation.
- High numerically stability – guaranteed convergence.
- Efficient solvers and parallelisation deliver unmatched productivity.
- Full 3D density gradient and 2D Poisson-Schrödinger quantum corrections
- Automatic generation of random discrete dopant, line edge roughness and material granularity.
- Advanced statistical reliability module.
- Integration with GSS Job Management tools and database data management enabling the use of large scale computing resources.
- Robust implementations of advanced Quantum correction methods for Monte Carlo simulation
- Direct particle-particle and particle-dopant interactions
- Fermi-Dirac statistics and implementation of Pauli's exclusion principle
- Generalised elastic/inelastic acoustic phonon modelling
- Generalised (polar-)optical phonon modelling
- Analytic multi-valley ellipsoidal non-parabolic conduction band model
- Inbuilt six-band k.p valence band model
- Import valence band structures from third party calculations
- Arbitrary user-defined analytic multi-valley band model
- Flexible band structure and scattering model definition, allowing custom materials and scattering models to be defined
- NEGF captures the wave-nature of the carriers
- Rigorous quantum mechanical treatment of tunnelling through NEGF

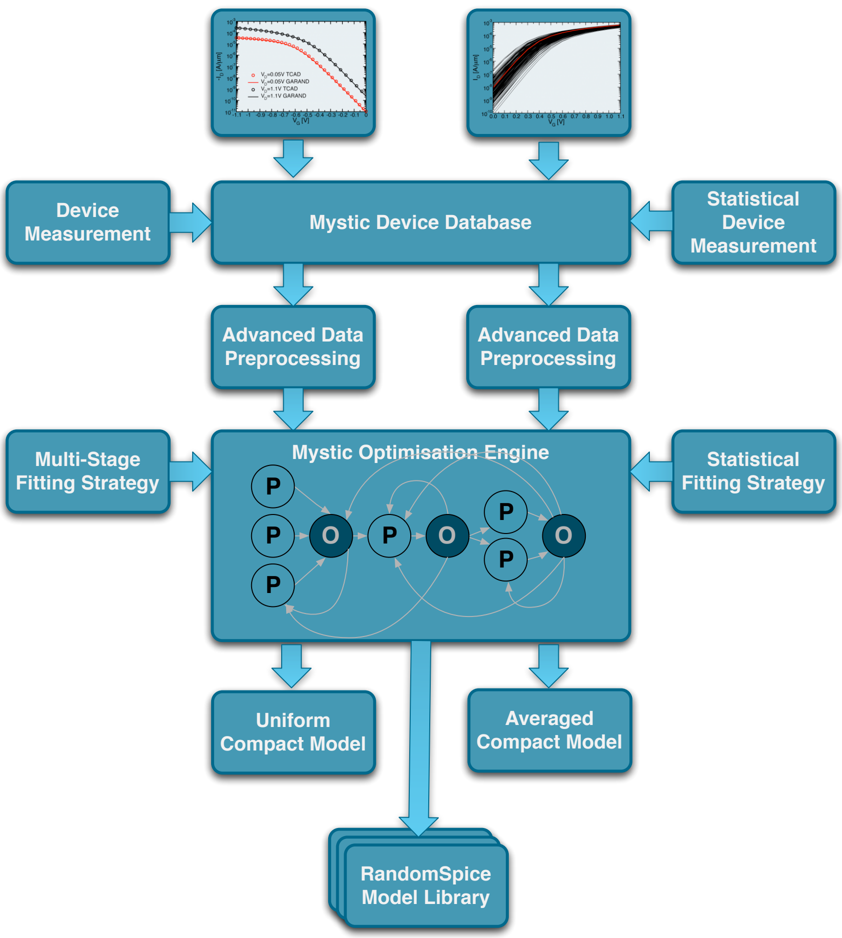
For further information please contact: *sales@GoldStandardSimulations.com*

Mystic

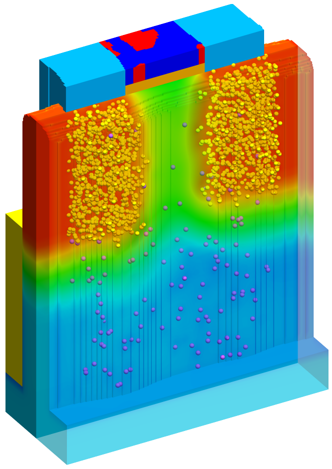
Advanced Statistical Compact Model Extractor

Product Description

Mystic is a state of the art compact model extraction tool that facilitates the extraction of compact model libraries, accurately capturing the interplay between process-induced variability and statistical variability and reliability in advanced technology generations. When combined with the advanced GSS ModelGEN technology, Mystic provides its users with the capability to accurately capture the highly non-Gaussian distributions and complex correlations between compact model parameters and transistor figures of merit, which are an absolute necessity for high-sigma circuit simulation and verification in a modern design workflow.

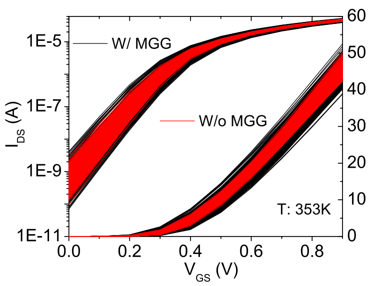


Unlike other compact model extraction tools, Mystic has been specifically designed to enable the rapid development and deployment of compact models in situations, such as transistor SRAM co-development in advanced CMOS technology, where the turnaround time from TCAD simulation or measurement to circuit simulation is critical to product and process development. When combined with the power and flexibility of the rest of the GSS tool chain, Mystic is the ideal tool for enabling and supporting Design/Technology Co-Optimization (DTCO).



Accurate physical simulation of statistical variability in 10nm CMOS FinFET including RDD, LER and MGG

Mystic provides a flexible and powerful scripted interface that allows the accurate extraction of advanced TCAD or Silicon-based compact models utilizing complex, highly accurate compact model extraction strategies.

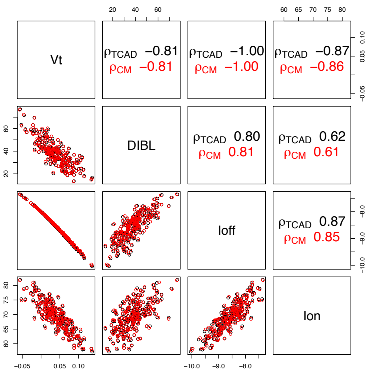


Simulated statistical set of I_D-V_G characteristics are targets for statistical compact model extraction

Mystic is fully integrated in the automated GSS tool flow and with the GSS data management system providing a seamless interface that allows users to easily manage and collate the thousands of data sets and models required for accurate statistical compact model extraction and circuit simulation.

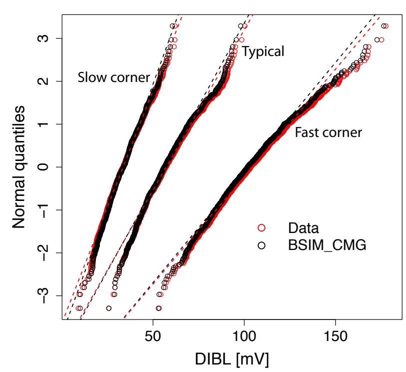
Based on carefully selected parameter sets and extraction strategies Mystic can accurately capture the interplay between:

- Process variability introduced by across wafer, wafer-to-wafer and lot-to-lot variations
- Purely statistical variability introduced by the discreteness of charge and granularity of matter
- Time dependent (BTI, HCI) variability introduced by charge trapping in the oxide and the generation of new defect states



Accuracy of extracted statistical compact models (red) against result of physical simulation (black)

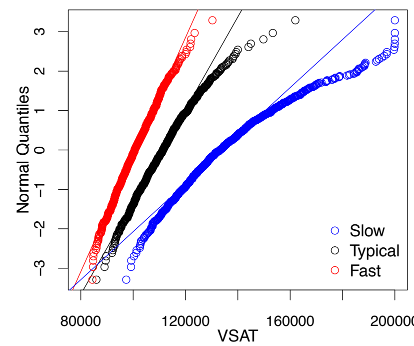
Mystic integration into the GSS tool-flow is enabled by a flexible Python interface which also provides access to a huge number of numerical and statistical library routines. This enables the adoption of a highly flexible approach to the analysis of statistical models from TCAD simulation or measurement, facilitating model QA and the analysis and validation of very large ensembles of statistical model cards.



The distribution of figures of merit from the extracted compact models (red) accurately reproduce the distributions from the physical simulations (black)

Using Mystic, users can generate highly flexible statistical compact model libraries that are capable of accurately modelling the behaviour of technology under a wide range of sources of process and statistical variability.

Mystic enables the production of GSS RandomSpice libraries utilising Look-Up Table (LUT), PCA, Gaussian parameter generators as well as supporting the advanced GSS ModelGEN technology, which delivers unprecedented accuracy in technology modelling for designers.



Typical non-gaussian BSIM-CMG parameter distributions from statistical compact model extraction at different process corners

Features

- Fast and accurate model extraction
- Flexible scripted interface
- A choice of 6 advanced model optimisation routines
- Full integration with the GSS data management framework
- Support for multiple spice simulators (HSpice, Eldo, Spectre, ngspice)
- Support for all major MOSFET models including BSIM-CMG
- Tailored for statistical model extraction
- Manage thousands of models with ease
- Customisable Database interface
- Customisable data output and analysis via Python
- Accurately capture the effects of NBTI and PBTI via time-dependant compact model extraction
- Full support for GSS ModelGEN, PCA and standard LUT and Gaussian generator based RandomSPICE libraries.
- Automatic generation of GSS ModelGEN libraries

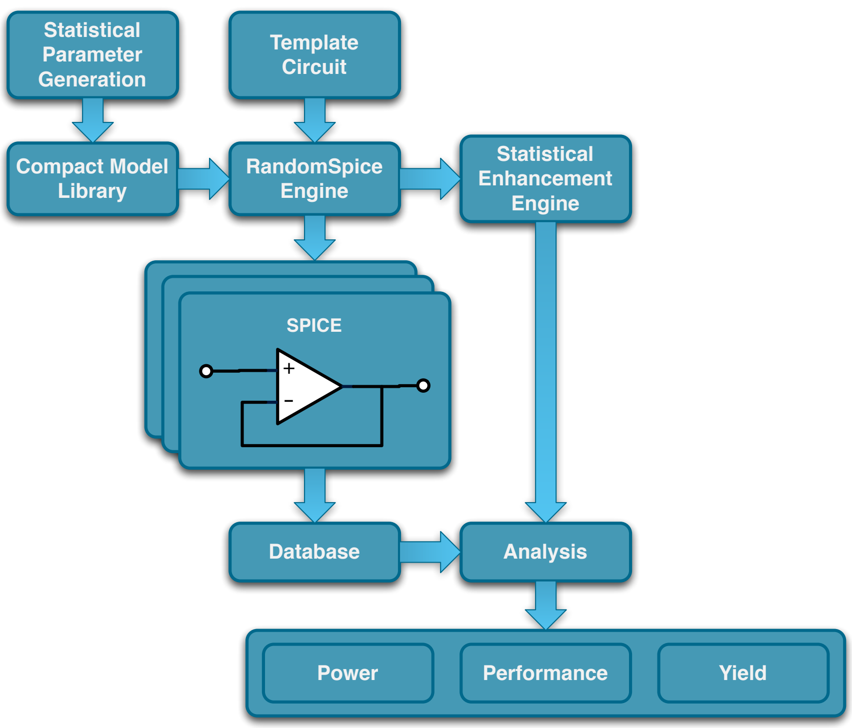
For further information please contact: *sales@GoldStandardSimulations.com*

RandomSpice

Statistical Circuit Simulator

Product Description

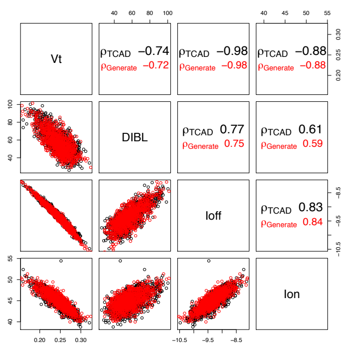
RandomSpice is an advanced statistical circuit simulation engine that incorporates a custom Monte Carlo-based technology that has been specifically designed to overcome the challenges of large-scale statistical circuit simulation, analysis and characterization needed to deal with the process and statistical variability issues prevalent in today's technologies. In concert with the GSS ModelGEN compact model generation technology RandomSpice delivers unparalleled accuracy in high-sigma simulation of modern designs utilising advanced technologies. RandomSpice provides a straightforward interface that allows the user to quickly and easily annotate a SPICE netlist for variability simulation and to generate extremely large ensembles of simulations using the SPICE backend of choice.



A highly flexible python data post-processing interface allows the user to intercept the simulation results "in-flight" and to calculate and analyse performance figures of merit of interest. This post-processing stage allows users to perform complex analyses on the very large amount of simulation data produced by statistical simulations in an efficient and intuitive manner. RandomSpice provides API's for direct integration with the GSS database and data management framework, allowing complex behavioural characterization to be performed across multiple simulation experiments and to conveniently consolidate the output simulation data in a single location for later analysis. This interface makes it trivial to perform power-performance-yield analyses, allowing fast turn-around during the design process and accurate performance margining.

RandomSpice is a highly efficient statistical simulation engine that has been tailored to facilitate the generation and execution of very-large-ensemble statistical circuit simulations. RandomSpice has been be designed to effectively operate on high performance computing clusters providing a rapid turnaround time when performing complex analyses of circuit performance in the presence of variability.

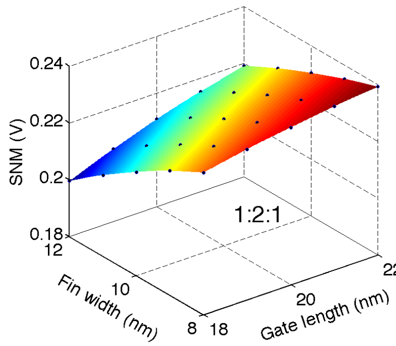
Through its integration with *Mystic*, RandomSpice harnesses the power of GSS ModelGEN technology to ensure that the complex shapes and correlations in extracted compact models are accurately captured.



Correlations of device figures of merit using ModelGEN Technology (red) in comparison with results from TCAD

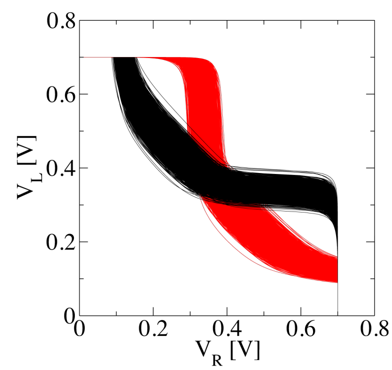
Highly non-Normal parameter distributions and strong correlations can be handled and accurately modelled to high-σ, allowing rare circuit configurations to be accurately assessed and the worst cases that dominate failure modes in high-density circuits such as SRAM to be fully investigated.

The combination of the computational and analysis capabilities of RandomSpice and ModelGEN facilitates users to accurately model the impact of many sources of variability on circuit performance, including process variability, statistical variability, reliability and degradation effects from N/PBTI and HCI. Via the power of ModelGEN's compact model generation algorithms RandomSpice provides users with the capability to generate effectively unlimited device ensembles with a combinations of variability and reliability effects in order to study lifetime circuit performance and to confidently predict product yield.

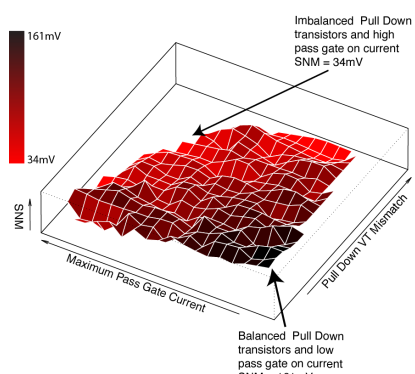


Impact of 10nm CMOS FinFET geometry on 1:1:1 SRAM cell static noise margin (SNM)

By leveraging the power of ModelGen, statistical enhancement and accelerated Monte Carlo techniques can be applied with confidence to enhance the computational efficiency of RandomSpice. The improved turn-around time allows designs to by proto-typed and analysed more quickly, reducing the design effort to improve performance and yield.

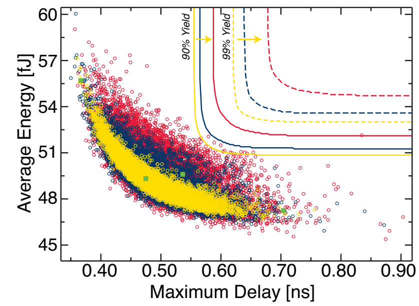


The impact of statistical variability on the SNM distribution of FinFET-based SRAM cells



Correlation between pull-down (PD) and pass (PA) transistor variability and SNM variability

RandomSpice accurately captures the correlation between process and statistical variability and allows performance-power-yield analysis and optimisation.



Power-Performance-Yield Analysis

Features

- Efficient statistical circuit simulation engine
- Introduce variability via simple annotation of SPICE netlists
- Support for multiple spice simulators (HSpice, Eldo, Spectre, ngspice)
- Support for all major MOSFET models including BSIM-CMG
- Designed for statistical circuit simulation
- Quick, easy analysis of Power-Performance-Yield trade-offs
- Easy deployment on HPC clusters
- Highly flexible python data post-processing
- Database API provides easy storage and analysis of very large datasets
- Full integration with the GSS data management framework
- Accurately model statistical variability
- Time dependent reliability model generators
- Process variability modelling
- Accurate high-sigma simulation
- Full integration of combined process, statistical and time-dependent variability
- GSS ModelGen compact model generators as well as PCA, LUT and Gaussian generators

For further information please contact: *sales@GoldStandardSimulations.com*