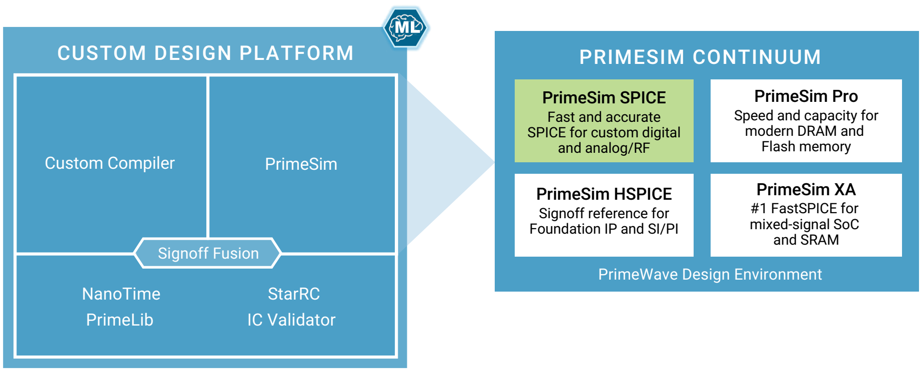
DATASHEET

PrimeSim SPICE

The world's fastest GPU-accelerated SPICE simulator for Analog, RF and mixed-signal design

Overview

PrimeSim SPICE is a high-performance SPICE circuit simulator for analog, RF, and mixed-signal applications. PrimeSim SPICE offers a unique multi-core/multi-machine scaling and heterogeneous compute acceleration on GPU/CPU delivering faster runtime with sign-off accuracy. PrimeSim SPICE supports High-frequency noise analysis, efficient S-Parameter handling while offering advanced analysis capabilities for periodic and non-periodic time-domain and frequency-domain applications. PrimeSim SPICE is fully integrated with PrimeWave Design Environment, a newly architected design environment and waveform viewer that encapsulates analysis and display requirements that significantly improve productivity. PrimeSim SPICE is a key engine inside PrimeSim Reliability analysis, a set of comprehensive solutions (Fault simulation, Static Circuit Check, MOS Aging, IR/EM, Variation) that analyze the full spectrum of a product life cycle, from early-life to end-of-life.



Benefits

- Industry-leading multi-core and multi-machine scaling and heterogeneous compute acceleration on GPU/CPU
- RF frequency domain and envelop analysis based on Shooting Newton and Harmonic Balance algorithms
- Capacity to handle large designs with hundreds of S-parameter ports for signal integrity applications
- Integrated safe operating area and other electrical rule checks
- Built on the golden HSPICE device model library, guaranteeing device model correlation between all PrimeSim engines
- Cloud-ready, built-in support containers for running the tool in a container as well as Synopsys cloud environment.

Superior Performance and Scalability

PrimeSim SPICE delivers the industry's best multi-core and multi-machine parallelization for massive scaling. With modern designs growing ever more complicated with an increased device and parasitic count, PrimeSim SPICE continues to deliver unmatched performance with the next-generation GPU-accelerated architecture which is uniquely geared towards simulating large designs with dense parasitic networks.

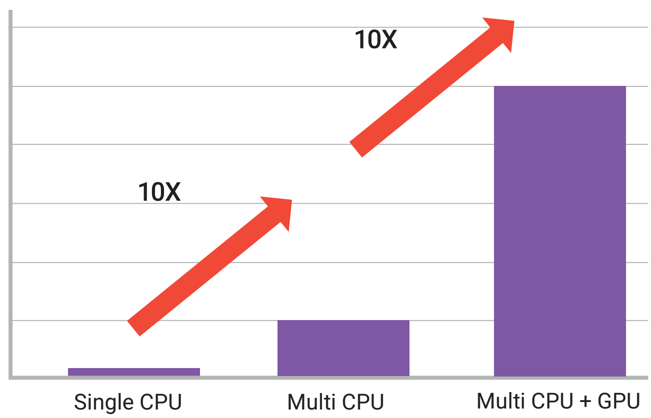


Figure 2: PrimeSim SPICE delivers superior scalability on CPU and GPU

Comprehensive analysis for analog, RF, and mixed-signal

For RF designs such as VCOs, LNAs, mixers, PLLs, and filters or analog and mixed-signal blocks such as data-converters, Delta-Sigma modulators, and switched-capacitor circuits, PrimeSim SPICE is a universal platform for all types of time domain, frequency domain, and noise analysis. The integration with the PrimeWave Design Environment provides powerful post-processing capabilities for measurements such as phase-noise, jitter, intermodulation distortion, gain compression, and conversion gain. PrimeSim SPICE is also integrated with the VCS digital simulator through a direct-kernel integration enabling high-performance mixed-mode simulation with SPICE, Verilog, SystemVerilog, Verilog-AMS, and VHDL.

ISO-26262 TCL-1 ASIL D Certified

- PrimeSim SPICE tool can be used in the development of safety-related elements according to ISO 26262, with allocated safety requirements up to a maximum Automotive Safety Integrity Level D (ASIL D), if the tool is used in the context of a tool chain and compliance of the PrimeSim SPICE Functional Safety Manual.

For more information about Synopsys products, support services or training, visit us on the web at: www.synopsys.com, contact your local sales representative or call 650.584.5000.