TRUE CIRCUITS INC™

JSPICE™

Summary

JSPICE™ is a powerful simulation environment developed and used by True Circuits over the last 25 years to create complex analog and digital circuits. JSPICE is now available for beta testing to a select group of users willing to provide valuable feedback in preparation for a full commercial release.

Features

Simulate with all foundry models
Simulate with Verilog-A, C or behavioral models
Run delay-accurate mixed-mode simulations
Perform logic/arithmetic synthesis
Perform Monte Carlo and timing analysis
Run millions of simulations in parallel
Cut design time to fraction
Configured for AWS™, Azure™, Google Cloud™
Experience the power of JSPICE environment

Includes

JSPICE Core Simulator

JSPICE Language Preprocessor
- Fully parameterized circuits
- C programming language
- Multi-dimensional buses
- Parameterized signal structures
- Directed logic/arithmetic synthesis
- MC and timing analysis/simulations
- Full industry SPICE compatibility
- User language extensions
- Compiled/encrypted libraries, and more

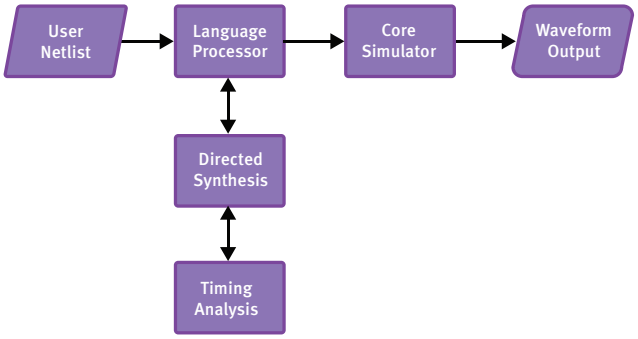
JSPICE Cloud Server
- Extremely high bandwidth
- Run simulations locally
- Run seamlessly on the cloud

Waveform analysis and sweep engine
Data analysis/reduction environment
Characterization with knowledge encapsulation
Schematic based electrical checks
General logic/arithmetic components libraries
Full documentation

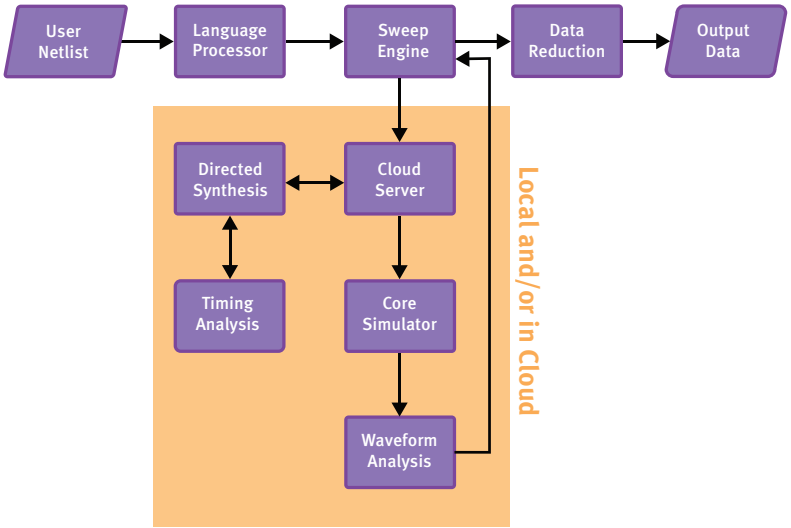
Typical User Scenarios

JSPICE makes it easy to express complicated circuits, run a large set of simulations in parallel, perform complex waveform analysis and reduce the results to a form easily understandable by the user, all leading to very rapid turnaround of circuit characterizations. Simulations can be run on local computers or in the cloud, or both as described below:

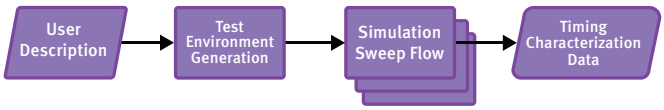
Single Simulation Flow



Simulation Sweep Flow



Characterization Flow



TRUE CIRCUITS INC™

JSPICE™

JSPICE Environment

The JSPICE environment makes it easy for users to rapidly design circuits. The key to rapid design is being able to quickly build intuition by iterating through changes and receiving quick and complete characterization feedback. JSPICE makes it easy to quickly define complex circuits, add structural options for testing, define insightful measurements, run complete characterizations over PVT and operating conditions in minutes, and automatically reduce the results to easily understood metrics that promote understanding and allow the user to make definitive circuit changes that avoid optimization of a flawed design. Closing this iteration loop in minutes rather than hours or days is instrumental to rapid design.

JSPICE Language

The JSPICE language is a dramatically extended version of generic SPICE circuit descriptions, which also includes optimized support for industry SPICE simulator syntax. While the language can be used with schematic capture, it essentially makes schematic capture obsolete for efficient design. The language adds both extended syntax and the C programming language to make statements surprisingly expressive and clear. It is compiled and supports precompiled objects for very fast access to large design libraries and data sets. It can generate not only generic SPICE output, but also Verilog, SDC constraints, LIB files, drive place and route flows for high-speed designs, and much more.

JSPICE Cloud Server

The JSPICE cloud server allows users to run single simulations or thousands in parallel with no additional effort. The server handles the task of scheduling simulation processes and transferring files and data to both local and cloud compute resources seamlessly. Local compute resources can be used efficiently for small jobs, while the cloud resources will be automatically allocated as needed within the user-predefined limits.

Beta Testing Program

Interested users, whether individuals, students or employees of companies, can submit an application by going to www.truecircuits.com/jspice.html. The application process will ask users to agree to the terms of the beta testing program, including providing periodic feedback and participating in user forums. Initially a small set of users will be selected by True Circuits, at its sole discretion, to begin beta testing for a selected period of time. This set will be expanded later. Approved users will be provided the JSPICE software suite, user guidelines, related documentation and a True Circuits point of contact for user support and feedback.

About True Circuits

True Circuits, Inc. offers a complete family of standardized, silicon-proven PLL and DLL hard macros that spans nearly all performance points and features typically requested by ASIC, FPGA and SoC designers. True Circuits utilizes robust state-of-the-art circuits, a methodical and proven design and test strategy, and close associations with the major foundries. True Circuits' PLL and DLL product portfolio is available in most TSMC, UMC and GLOBALFOUNDRIES processes and process variants from 180nm to 4nm.

Since 1998, True Circuits has distinguished itself as the technology leader in the timing IP space, and its PLLs and DLLs are used extensively around the world in its customers' products with production volumes in the billions.