

PSpice vs. Free Board Level Analog Simulator

Feature Comparison Matrix

	Free Simulator	PSpice® Technology
Libraries and Modeling		
Models	Primarily LT device	Several leading device and component manufacturers
Simulation model shipped with installation	~2000 models	~34000+ models
Model editing and new model creation		•
Modeling apps		•
Devices		
Advanced magnetics		•
IGBT		•
True digital primitives		•
RAM/ROM		•
GaAsFET Model - TOM-2 and TOM-3		•
Coupled transmission line		•
Jiles-Atherton magnetic core model		•
Simulation		
Solvers	Multiple	Multiple
Automatic convergence diagnostic and resolution tool		•
Advanced convergence aids	Limited capability	Exhaustive
Checkpoint/restart		•
Resume/continue mode		•
Accuracy vs. speed trade-off	Limited capability	Automated - SPEED dial
User-defined functions	•	•
Assertions		•
Simulation manager - schedule/manage set of simulations		•
Waveform Analysis		
Axis settings	Limited capability	Exhaustive
Support multiple trace plot as one (plot template)		•
Support text and waveform labeling		•
Performance analysis		•
Plot histogram		•
Perform measurement with waveform analysis		•
Post-analysis measurement		
Cursor support for multiple traces/plot	Limited capability	•
Pre-configured, commonly used one-click cursor navigation		•
Waveform arithmetic	•	•
User-defined expression/macro	•	•
Export simulation result	•	•
Create stimulus from simulation result		•
Ability to plot B-H curve		•
Hysteresis core loss calculation		•
Marching waveform	•	•
Compare different simulation results - one on top of other		•
Read data from oscilloscope		•
Ability to annotate operating point result on schematic		•
Support for scripting language (Tcl)		•

	Free Simulator	PSpice® Technology
Encryption		
Basic	•	•
AES		•
DES		•
Personal key-based encryption		•
Mixed-Mode Simulation		
Event-based simulator		•
Ability to model timing characteristics of digital devices		•
Ability to model I/O characteristics of digital devices		•
Programmable logic arrays		•
Random access memory		•
Tri-state gates		•
Digital stimulus		•
Logic expression		•
Pin-to-pin delay		•
Constraint		•
EDGE and level-triggered devices		•
SETUP/HOLD timing		•
Auto insertion of AtoD and DtoA devices		•
Design and Modelling Utilities		
Model Editor - extract model parameter from datasheet curves		•
Stimulus Editor		•
Magnetics design, modeling, and manufacturing		•
Categorized libraries		•
Search based on parametric data of devices		•
Report generation		•
Schematic Capture		
Multi-sheet design		•
Symbol generation	Limited capability	Comprehensive wizard-based approach
Archive project		•
Integrated SI simulation environment		•
Integrated PCB tools		•
Integrated component information system solution/flows		•
Design reuse (start a new design from existing design)		•
In-schematic modeling	Very limited	•
System Design Capability		
Integration with MATLAB - Simulink		•
Goal-based circuit optimization		•
Waveform-based circuit optimization		•
Sensitivity analysis		•
Analog worst-case analysis		•
Digital worst-case timing analysis		•
C/C++ based model development and simulation		•
Verilog-A model simulation		•
Stress analysis (SMOKE)		•
Heat sink simulation and component deration design		•
Design space exploration		•
Monte Carlo analysis	Limited	•
Miscellaneous		
Learning module		•
Design template for most commonly used circuits		•
Design examples	•	•
Help	Limited - only offline	Exhaustive, both online and offline
Support	?	•
Updates	Unplanned	Regular and has predictable schedule
New features and enhancements	?	•