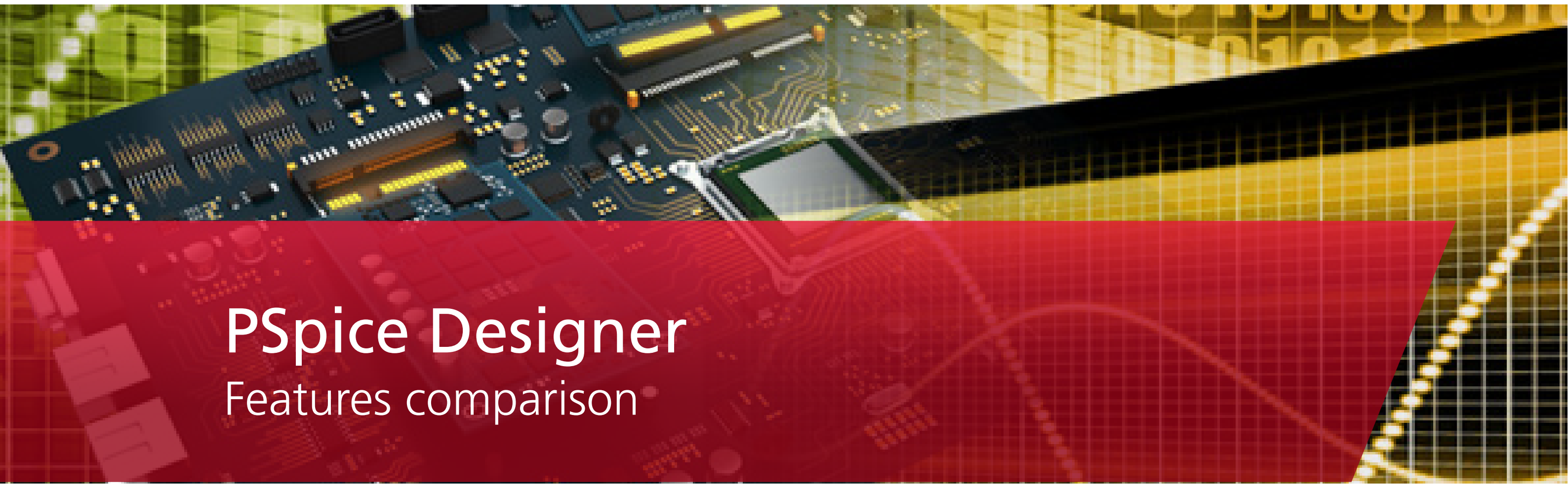

PSpice Designer

Features comparison

Feature	OrCAD PSpice Basic	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
OrCAD CAPTURE SCHEMATIC ENTRY + COMPONANTS DATABASE MANAGEMENT	Not Applicable		
Graphical, flat and hierarchical page editor and Picture block hierarchy		•	•
Unlimited undo/redo		•	•
Net Groups - Complex bus definition		•	•
Intelligent PDF creation		•	•
AutoWire		•	•
Design reuse		•	•
3D Footprint Viewer		•	•
Coloured Components / nets		•	•
Tcl scripting customization		•	•
Online design rule check including custom DRC capability and Waive DRC		•	•
PCB Forward and back-annotation of properties / pin-and-gate swaps		•	•
Schematic Part and Library editor		•	•
Cross-probing and cross-placement		•	•
FPGA design-in / pin import & export and FPGA bi-directional support		•	•
Multiple PCB netlist interfaces		•	•
SI Topology creation		•	•
Digi-Key (PartLink App) Component Parametric data directly from web		•	•
Property editor for pins, components, nets		•	•

• Fonction included

Feature	OrCAD PSpice Basic	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
Design differences viewer		•	•
Intelligent PDF creation		•	•
Component Information System			
Centralized part information system		CIS option	•
Relational data support		CIS option	•
ODBC-compliant database support		CIS option	•
Graphical preview of database parts		CIS option	•
Intelligent database query		CIS option	•
Component property validation		CIS option	•
Interface to relational database and management systems		CIS option	•
Database query for part selection and parametric properties		CIS option	•
Extensive reports and report templates		CIS option	•
Crystal Reports™ for advanced documentation		CIS option	•
Unlimited assembly variant support		CIS option	•
Schematic and BOM Variants Manager (Parts not Fitted and more).		CIS option	•
CIS Database Management Interface (access control and more)		CIS Option + CIP E Option	CIP E Option
Part search DIGIKEY, FARNELL, FUTURE, MOUSER, ARROW		CIS option	•
Import			
PSpice schematic, EDIF, PDIF, XML		•	•
PADS schematic design		•	•
Import Altium schematic design		•	•
Import Eagle schematic design		•	•
Export			
PDF, DXF,EDIF, XML, ISCF		•	•
• PSPICE SIMULATION			
Simulation			
Maximum number of nodes and devices in the design	250 nodes or 250 devices	No limits	No limits
DC sweep & AC sweep analysis	10,000 data points	No limits	No limits
Transient analysis	1,000,000 data points	No limits	No limits
SPICE Monte Carlo Analysis	•	•	•

• *Fonction included*

Feature	OrCAD PSpice Basic	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
SPICE Sensitivity Analysis	•	•	•
SPICE Worst Case Analysis	•	•	•
SPICE Parametric Sweep analysis	•	•	•
Temperature sweep analysis	•	•	•
Checkpoint/Restart analysis	•	•	•
Advanced convergence control / options	•	•	•
Analog behavioral modeling	•	•	•
Interactive waveform viewer & analyzer	•	•	•
Auto-convergence	•	•	•
Partial design simulation	•	•	•
Advanced control option	•	•	•
Multi-core engine support	•	•	•
Tcl customization for custom analysis / post-processing	•	•	•
Expression support for post-processing	•	•	•
Digital Worst Case with built-in 6 levels support	•	•	•
Frequency Response Analysis	•	•	•
PSpice Reports	•	•	•
Models			
Stimulus editor		•	•
33,000+ simulation-ready parts	•	•	•
BSIM 3.3 & BSIM 4 devices		•	•
Magnetic core		•	•
IGBT		•	•
Tlines		•	•
DML model support		•	•
IBIS model support		•	•
Model Editor for device characterization		•	•
Model development using PSpice Device Model Interface		•	•
Magnetics Part Editor		•	•
Library Encryption using AES 256 bit algorithm	•	•	•
Waveform Analysis			
Measurement	•	•	•
Performance Analysis		•	•
Advanced Tools (FRA, Core loss)	•	•	•

• *Fonction included*

Feature	OrCAD PSpice Basic	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
FFT	•	•	•
PSpice Advanced Analysis (PAA)			
Smoke: Detects component stress		•	•
Advanced Analysis			•
Advanced Sensitivity: Identifies critical circuit components			•
Optimizer: Optimizes key circuit components			•
Optimizer: Curve fitting			•
Advanced Monte Carlo: Analyzes statistical circuit behavior and yield			•
Parametric Plotter: Solution analysis through nested sweeps			•
PSpice Device Model Interface Model Simulation			•
PSpice System Option			•
PSpice Systems Option			
Simulate electrical circuits and mechanical/hydraulic/ thermal blocks together			•
Simulate with ideal models for faster simulation during proof of concept			•
Simulate with actual electrical designs using PSpice component models			•
Electrical simulations with PSpice models exhibit nonlinearities, delay, and other real-world effects			•
Large library of electrical parts for PSpice simulation and mechanical models and pre-defined blocks for Simulink available			•
Full access to PSpice environment for in-depth electrical design and debugging			•
Full access to MATLAB for analyzing and visualizing data, developing graphical user interfaces, and creating model data and parameters			•

• Fonction included

1 bis, avenue Foch – 94100 Saint-Maur – France – Tél.: +33 (0)1 77 01 82 90 – info@artedas.eu – **www.artedas.eu**