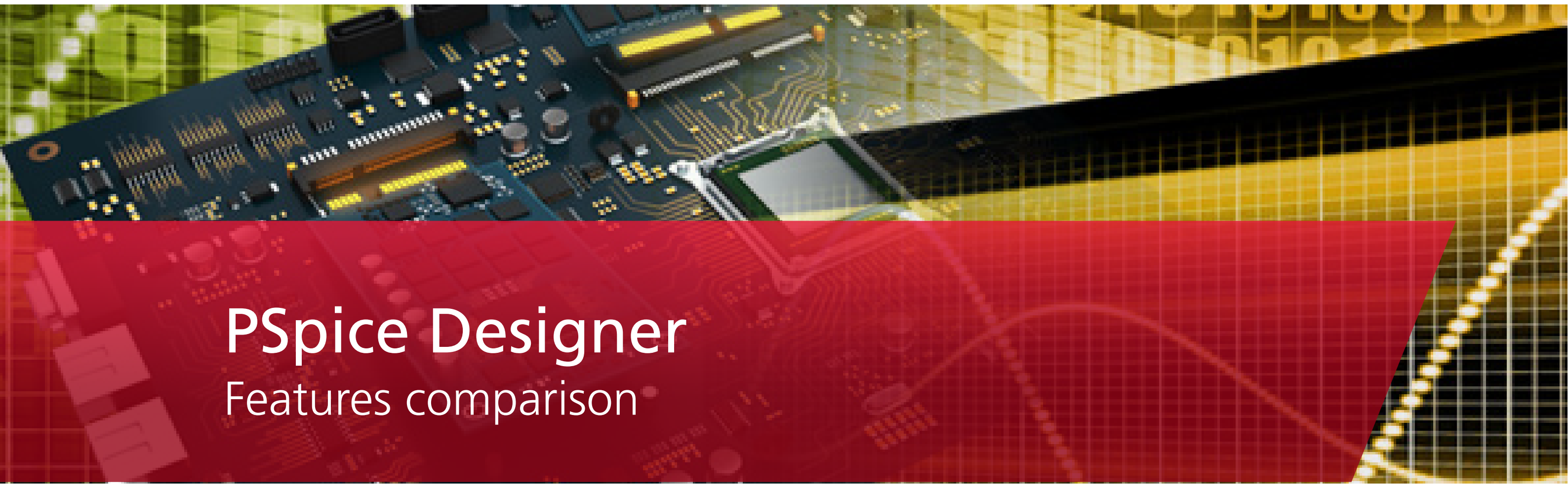
cadence® OrCAD™ CADENCE EDA SOLUTIONS



PSpice Designer

Features comparison

Feature	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
OrCAD CAPTURE SCHEMATIC ENTRY + COMPONANTS DATABASE MANAGEMENT		
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 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	No limits	No limits
DC sweep & AC sweep analysis	No limits	No limits
Transient analysis	No limits	No limits
SPICE Monte Carlo Analysis	•	•
SPICE Sensitivity Analysis	•	•
SPICE Worst Case Analysis	•	•
SPICE Parametric Sweep analysis	•	•
Temperature sweep analysis	•	•

• *Fonction included*

Feature	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
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)	•	•
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		•

• *Fonction included*

Feature	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
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**

15-01-2021