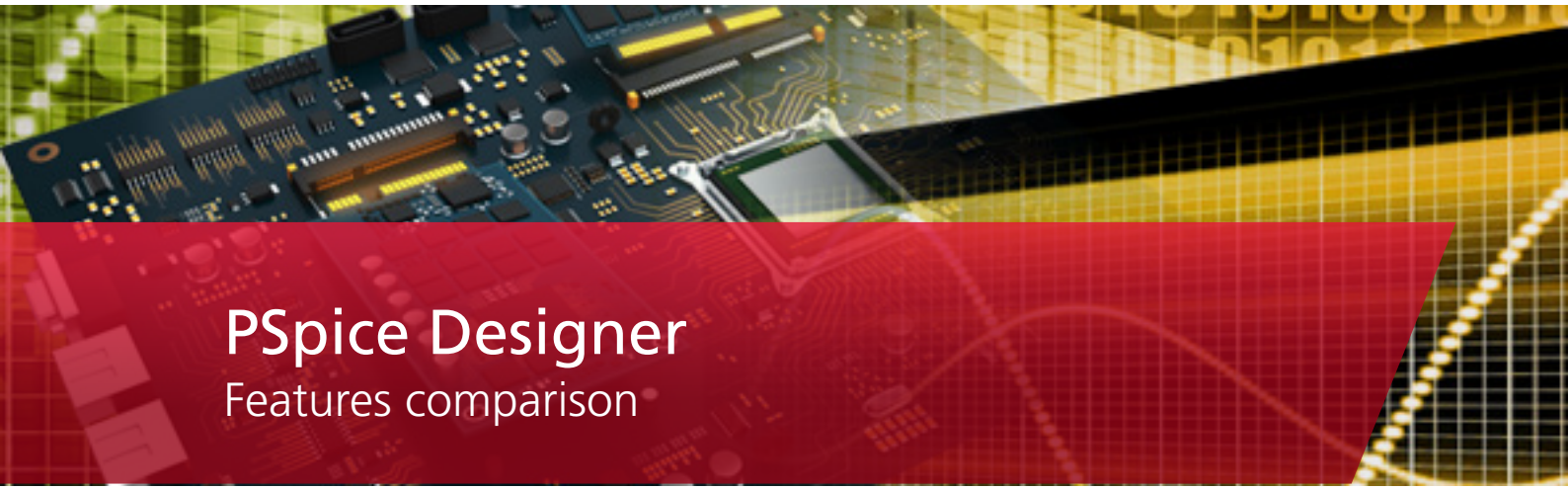
cadence® OrCAD™
CADENCE EDA SOLUTIONS



PSpice Designer
Features comparison

Feature	OrCAD PSpice Designer Standard	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 Designer Standard	OrCAD PSpice Designer	OrCAD PSpice Designer Plus
Design differences viewer		•	•
Intelligent PDF creation		•	•
Component Information System			
Centralized part information system		CIS option	CIS option
Relational data support		CIS option	CIS option
ODBC-compliant database support		CIS option	CIS option
Graphical preview of database parts		CIS option	CIS option
Intelligent database query		CIS option	CIS option
Component property validation		CIS option	CIS option
Interface to relational database and management systems		CIS option	CIS option
Database query for part selection and parametric properties		CIS option	CIS option
Extensive reports and report templates		CIS option	CIS option
Crystal Reports™ for advanced documentation		CIS option	CIS option
Unlimited assembly variant support		CIS option	CIS option
Schematic and BOM Variants Manager (Parts not Fitted and more).		CIS option	CIS option
CIS Database Management Interface (access control and more)		CIS Option + CIP E Option	CIS Option + CIP E Option
Part search DIGIKEY, FARNELL, FUTURE, MOUSER, ARROW		CIS option	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 Designer Standard	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 Designer Standard	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**

23-11-2020