

# OrCAD Professional Plus – POX300

## Product Description

### OrCAD X Professional Plus (POX300)

**OrCAD Professional Plus (POX300)** includes everything in **OrCAD Professional (POX200)**, with the addition of *PSpice's* full functionality, including *Advanced Analysis* and *Systems Option*.

Engineers can maximize design performance, yield, cost-effectiveness, and reliability with *PSpice Advanced Analysis* used in conjunction with core *PSpice*. *PSpice Advanced Analysis* addresses engineers' concerns about testing designs to meet real environmental constraints. Now, engineers will know how their designs perform with manufacturing tolerance variations, external temperature, operational range, and aging components. Engineers can test components when they are stressed—know how designs perform in the field, whether there are any critical sensitivities, and what might fail during manufacturing.

**OrCAD Professional Plus** also includes ***PSpice Advanced Option***, so now engineers can:

- Improve design quality and productivity with automatic performance optimization algorithms
- Find optimum combinations of component values, automatically simulate, evaluate results, and adjust component values to reach performance requirements
- Retarget existing designs with new goals without modifying the layout
- Go beyond debugging waveforms— optimize design, improve design reliability and manufacturing yield, and reduce the cost of components
- Improved accuracy with tight integration to schematic tools

**OrCAD Professional Plus** also includes ***PSpice Systems Option***, so now engineers can:

- Perform system-level simulation powered by PSpice and MathWorks Simulink
- Cadence PSpice MATLAB Simulink Interface combines two industry-leading simulation tools to create an environment for electro-mechanical system simulation

**OrCAD Professional Plus** also includes ***PSpice System Analysis***, so now engineers can:

- Co-simulate Simulink and PSpice together for complete system simulation
- View PSpice simulation results in MATLAB
- Use MATLAB functions directly in measurement expressions within PSpice
- Export behavioral models from Simulink to PSpice using Cadence's Device Modeling Interface (DMI)

