

## OrCAD for Education Program

In partnership with Cadence Design Systems (the world's largest EDA software vendor), ECADtools administers the **OrCAD for Education Program**, which provides universities, electronics engineering students, academic staff and researchers access to Cadence's best-of-breed PCB design and simulation tools, – all at ultra-low prices.

In November 2016, Cadence also announced ultra-low priced, fully-functional design and simulation software for students; with the same \$29,000+ design suite available to universities now available to students for just \$125 per academic year.

These bundles are feature-rich, packed with the same capabilities as OrCAD's commercial packages; including:

1. embedded learning resources within the apps, including tutorials and design exercises
2. the ability to add your own course materials (as HTML documents) into the apps, via an open API
3. access to more than 12,000 FREE reference designs (from SiliconExperts)
4. access to more than 35,000 FREE PSpice models (see more at pspice.com)
5. access to online component libraries from companies such as DigiKey and Arrow

The following matrix shows the features of each University Edition bundled suite, with pricing options for universities and students.

FEATURE	Freeware	Education Packages	
	OrCAD 'Lite'	OrCAD 'Simulation' (Prod Code: PO150E)	OrCAD 'Design + Simulation' (Prod Code: PO250E)
<b>Design tools</b>			
<b>OrCAD 'Capture'</b> (Schematic entry)	Limited	✓	✓
<b>OrCAD 'PCB Editor'</b> (Advanced PCB layout)	Limited	X	✓
<b>Simulation tools</b>			
<b>PSpice 'A/D'</b> (Mixed signal SPICE simulator)	Limited	✓	✓
<b>PSpice 'AA'</b> (Advanced Analysis)	Limited	✓	✓
<b>PSpice 'SLPS'</b> (Simulink/PSpice co-simulation)	Not Included	✓	✓
<b>PSpice 'Systems Option'</b> (PCB signal integrity checker)	Not Included	✓	✓
<b>OrCAD 'PCB SI'</b> (PCB signal integrity checker)	Limited	x	✓
<b>Data Management tools</b>			
<b>OrCAD 'CIS'</b> (Component data management)	Limited	✓	✓
<b>Commercial Value</b>		>\$31,000	>\$39,000
<b>UNIVERSITY PRICING</b>			
	<b>Rent:</b>	<b>\$125</b> pa	<b>\$149</b> pa
	<b>Own:</b>	<b>\$410</b>	<b>\$559</b>
<b>STUDENT PRICING</b>			
	<b>Rent:</b>	NA	<b>\$125</b> pa

#### Prices:

- Australian Dollars (AU\$). Include 10% GST
- Include software update service for one year

#### Features:

- ✓ Indicates included features

## Program benefits

### Why Cadence?

- At ~\$1.9Billion pa, they are the largest PCB and IC design and simulation software vendor, turning over annually more than twice that of Matlab, Zuken and Altium combined.
- Invest more than \$600Mill pa in new product development.
- OrCAD design suites provide an integrated 'design + simulation + component data management' workspace and are not a set of discrete, non-integrated apps, thus enabling rapid, error-free transition between functions and customizable workflows.
- PSpice is the most powerful SPICE simulation and design analysis tool on the market.
- Only Cadence has partnered with Mathworks to deliver a range of MATLAB Simulink/PSpice 'co-simulation' tools for advanced 'system + electrical' simulation.
- Students proficient in OrCAD and Pspice are in demand in Asia, India, North America & the UK.

### Easy to join

- If you're an educational institution delivering PCB design courses, you can participate simply by buying or leasing a minimum of twenty (20) licences. That's as little as AU\$2,500 per annum.
- Simply decide on the package and licence options that meet your needs.

**Discover more:**      [www.ecadtools.com.au/orcad-for-universities.html](http://www.ecadtools.com.au/orcad-for-universities.html)

### Amazing Benefits

- Free resources include:
  - o 12,000+ reference designs
  - o 35,000+ PSpice component models shipped with the app, plus 1,000s more free PSpice models online from vendors such as TI, Renesas, STMicro, etc
  - o component data libraries
  - o 100s of how-to video tutorials

### Student Support

- Students studying at a participating educational institution:
  - o Can lease '**OrCAD PCB Designer - Student Edition**' for personal use for just **AU\$125** per annum (GST and software update service included)
  - o Conditions are simple:
    - Must apply via their institutional email address (@uniname.edu.au in Australia; @uniname.ac.nz in New Zealand) to confirm they are enrolled at a participating institution.
    - One licence per student per year, locked to a personal computer, not transferrable.
    - Purchases can be made online in the first 4 weeks of each new term.

**Discover more:**      [www.ecadtools.com.au/orcad-for-students.html](http://www.ecadtools.com.au/orcad-for-students.html)

### Local Support

- Locally supported by ECADtools, Sydney.

## OrCAD 'Simulation' features

FEATURE	'Lite'	PO150E
<b>Capture - Schematic Entry</b>		
Graphical, flat and hierarchical design	LIMITED	✓
Heterogeneous bus / NetGroup support	LIMITED	✓
Unlimited undo/redo	LIMITED	✓
Dynamically update hierarchical blocks	LIMITED	✓
Design reuse	LIMITED	✓
Reference external designs / circuits	LIMITED	✓
Tcl customization	LIMITED	✓
Online design rule check	LIMITED	✓
Unlimited user-defined properties	LIMITED	✓
Support heterogeneous / homogenous parts	LIMITED	✓
PCB forward- and back-annotation	LIMITED	✓
Schematic part and library editor	LIMITED	✓
44,000 Schematic symbols FREE online	LIMITED	✓
Colour components / nets	LIMITED	✓
PCB cross-probing and cross-placement	LIMITED	✓
FPGA design-in support	LIMITED	✓
FPGA bi-directional support	LIMITED	✓
Intelligent PDF creation	LIMITED	✓
Flow support for most PCB layout tools	LIMITED	✓
Property editor for pins, components, nets	LIMITED	✓
DigiKey Component Parametric data directly from web	LIMITED	✓
Export to PDF, DXF, EDIF, XML, ISCF	LIMITED	✓
Design differences viewer	LIMITED	✓
Import PSpice schematic, EDIF, PDIF, XML	LIMITED	✓
Import PADS schematic design	LIMITED	✓
Import Altium schematic design	LIMITED	✓
<b>CIS - Component Information Data Management System</b>		
Centralized part information system	LIMITED	✓
ODBC-compliant database support	LIMITED	✓
Relational data support	LIMITED	✓
MRP, ERP, and PLM integration	LIMITED	✓
Graphical preview of database parts	LIMITED	✓
Intelligent database query	LIMITED	✓
Component property validation	LIMITED	✓
Temporary / new part introduction	LIMITED	✓
Extensive reports / report templates	LIMITED	✓
Crystal reports for advanced documentation	LIMITED	✓

OrCAD 'Simulation' features		
Feature	'Lite'	PO150E
<b>Assembly Variants</b>		
Unlimited Assembly variant support	LIMITED	✓
Part substitution and part "not present" support	LIMITED	✓
Variant comparison reports	LIMITED	✓
Print capabilities for variants	LIMITED	✓
<b>PSpice Analogue/Digital (A/D)</b>		
DC sweep, AC sweep, & transient analysis	LIMITED	✓
SPICE Monte Carlo Analysis	LIMITED	✓
SPICE Sensitivity Analysis	LIMITED	✓
SPICE Worst Case Analysis	LIMITED	✓
SPICE Parametric Sweep analysis	LIMITED	✓
Temperature sweep analysis	LIMITED	✓
Checkpoint/Restart analysis	LIMITED	✓
Advanced convergence control / options	LIMITED	✓
Analogue behavioural modelling	LIMITED	✓
Stimulus editor	LIMITED	✓
Model Editor for device characterization	LIMITED	✓
Magnetics Part Editor	LIMITED	✓
Library Encryption using AES 256 bit algorithm	✓	✓
Interactive waveform viewer & analyser	LIMITED	✓
Auto-convergence	LIMITED	✓
Partial design simulation	LIMITED	✓
Advanced control option	LIMITED	✓
Multi-core engine support	LIMITED	✓
Tcl customization for custom analysis / post-processing	LIMITED	✓
33,000+ simulation-ready part models FREE	✓	✓
Expression support for post-processing	LIMITED	✓
Digital Worst Case with built-in 6 levels support	LIMITED	✓
IBIS / DML model support	LIMITED	✓
Frequency Response Analysis	LIMITED	✓
PSpice Reports	LIMITED	✓
Model development using PSpice Device Model Interface	NA	✓
<b>PSpice AA (Advanced Analysis)</b>		
Advanced Sensitivity: Identifies critical circuit components	LIMITED	✓
Optimiser: Optimises key circuit components	LIMITED	✓
Optimiser: Curve fitting	LIMITED	✓
Advanced Monte Carlo: Analyses statistical circuit behaviour and yield	LIMITED	✓
Smoke: Detects component stress	LIMITED	✓
Parametric Plotter: Solution analysis through nested sweeps	LIMITED	✓
PSpice Device Model Interface Model Simulation	LIMITED	✓
<b>PSpice SLPS (Simulink/PSpice Co-Simulation)</b>		
MATLAB Simulink / PSpice system/electrical modelling integration	NA	✓
<b>PSpice Systems Option (bi-directional data handling)</b>		
New Product	NA	✓

## OrCAD 'Design + Simulation' features

Feature	'Lite'	PO250E
<b>Capture - Schematic Entry</b>		
Graphical, flat and hierarchical design	LIMITED	✓
Heterogeneous bus / NetGroup support	LIMITED	✓
Unlimited undo/redo	LIMITED	✓
Dynamically update hierarchical blocks	LIMITED	✓
Design reuse	LIMITED	✓
Reference external designs / circuits	LIMITED	✓
Tcl customization	LIMITED	✓
Online design rule check	LIMITED	✓
Unlimited user-defined properties	LIMITED	✓
Support heterogeneous / homogenous parts	LIMITED	✓
PCB forward- and back-annotation	LIMITED	✓
Schematic part and library editor	LIMITED	✓
44,000 Schematic symbols FREE online	LIMITED	✓
Colour components / nets	LIMITED	✓
PCB cross-probing and cross-placement	LIMITED	✓
FPGA design-in support	LIMITED	✓
FPGA bi-directional support	LIMITED	✓
Intelligent PDF creation	LIMITED	✓
Flow support for most PCB layout tools	LIMITED	✓
Property editor for pins, components, nets	LIMITED	✓
DigiKey Component Parametric data directly from web	LIMITED	✓
Unlimited Assembly variant support	LIMITED	✓
Part substitution and part "not present" support	LIMITED	✓
Variant comparison reports	LIMITED	✓
Print capabilities for variants	LIMITED	✓
Export to PDF,DXF, EDIF, XML, ISCF	LIMITED	✓
Design differences viewer	LIMITED	✓
Import PSpice schematic, EDIF, PDIF, XML	LIMITED	✓
Import PADS schematic design	LIMITED	✓
Import Altium schematic design	LIMITED	✓
<b>CIS - Component Information Data Management System</b>		
Centralised part information system	LIMITED	✓
ODBC-compliant database support	LIMITED	✓
Relational data support	LIMITED	✓
MRP, ERP, and PLM integration	LIMITED	✓
Graphical preview of database parts	LIMITED	✓
Intelligent database query	LIMITED	✓
Component property validation	LIMITED	✓
Temporary / new part introduction	LIMITED	✓
Extensive reports / report templates	LIMITED	✓
Crystal Reports for advanced documentation	LIMITED	✓

## OrCAD 'Design + Simulation' features

Feature	'Lite'	PO250E
<b>PSpice A/D (Analogue/Digital)</b>		
DC sweep, AC sweep, & transient analysis	LIMITED	✓
SPICE Monte Carlo Analysis	LIMITED	✓
SPICE Sensitivity Analysis	LIMITED	✓
SPICE Worst Case Analysis	LIMITED	✓
SPICE Parametric Sweep analysis	LIMITED	✓
Temperature sweep analysis	LIMITED	✓
Checkpoint/Restart analysis	LIMITED	✓
Advanced convergence control / options	LIMITED	✓
Analogue behavioural modelling	LIMITED	✓
Stimulus editor	LIMITED	✓
Model Editor for device characterization	LIMITED	✓
Magnetics Part Editor	LIMITED	✓
Library Encryption using AES 256 bit algorithm	✓	✓
Interactive waveform viewer & analyser	LIMITED	✓
Auto-convergence	LIMITED	✓
Partial design simulation	LIMITED	✓
Advanced control option	LIMITED	✓
Multi-core engine support	LIMITED	✓
Tcl customization for custom analysis / post-processing	LIMITED	✓
33,000+ simulation-ready part models FREE	✓	✓
Expression support for post-processing	LIMITED	✓
Digital Worst Case with built-in 6 levels support	LIMITED	✓
IBIS / DML model support	LIMITED	✓
Frequency Response Analysis	LIMITED	✓
PSpice Reports	LIMITED	✓
Model development using PSpice Device Model Interface	NA	✓
<b>PSpice AA (Advanced Analysis)</b>		
Advanced Sensitivity: Identifies critical circuit components	LIMITED	✓
Optimiser: Optimises key circuit components	LIMITED	✓
Optimiser: Curve fitting	LIMITED	✓
Advanced Monte Carlo: Analyses statistical circuit behaviour and yield	LIMITED	✓
Smoke: Detects component stress	LIMITED	✓
Parametric Plotter: Solution analysis through nested sweeps	LIMITED	✓
PSpice Device Model Interface Model Simulation	LIMITED	✓
<b>PSpice SLPS (Simulink/PSpice Co-Simulation)</b>		
MATLAB Simulink / PSpice system/electrical modelling integration	NA	✓
<b>PSpice Systems Option (bi-directional data handling)</b>		
New Product	NA	✓

## OrCAD 'Design + Simulation' features

Feature	'Lite'	PO250E
<b>PCB Editor</b>		
Place application mode	LIMITED	✓
Cross-place and cross-probe with OrCAD Capture	LIMITED	✓
3D visualization / flip-board	LIMITED	✓
Quickplace utility	LIMITED	✓
Component alignment	LIMITED	✓
Place by room / by schematic page	LIMITED	✓
Placement replication	LIMITED	✓
Native 3D viewer (STEP & Simple extrusions)	LIMITED	✓
Floor-planning, Auto-place	LIMITED	✓
Component height checks	LIMITED	✓
Etch edit app mode	LIMITED	✓
Interactive routing	LIMITED	✓
Scribble route	LIMITED	✓
Group routing (Multi-line routing)	LIMITED	✓
Diff pair routing	LIMITED	✓
Snake routing (Hex pitch BGA)	LIMITED	✓
Jumper support (single-sided design)	LIMITED	✓
Slide/Split stacked vias	LIMITED	✓
Fan-out generators	LIMITED	✓
Copy fanout to same packages	LIMITED	✓
Single click multiple hdi-via instantiation	LIMITED	✓
Dynamic ratnest suppression during Routing	LIMITED	✓
Spread lines between anti-pads	LIMITED	✓
Segment over void detection	LIMITED	✓
Contour routing - flex	LIMITED	✓
Interactive delay tuning	NA	✓
Dynamic heads-up display	LIMITED	✓
Shape based curve fillet support, tapered traces	NA	✓
<b>Rigid Flex PCB Editor</b>		
Cross Section support for mask/coating layers	LIMITED	✓
Native database flex and surface finish layers	LIMITED	✓
Auto-rounding shape corners	LIMITED	✓
Dynamic cross hatch and solid planes	LIMITED	✓
Cross section by zone	LIMITED	✓
Inter layer DRC (mask to mask; mask to surface conductor)	LIMITED	✓
Zone table chart for manufacturing	LIMITED	✓
Dynamic zone placement (auto drop down)	LIMITED	✓
Techfile support of zones	LIMITED	✓
Curved fillets	LIMITED	✓

## OrCAD 'Design + Simulation' features

Feature	'Lite'	PO250E
<b>Auto-Routing</b>		
6-signal-layer autorouting	LIMITED	✓
SMD fanout	LIMITED	✓
Trace width by net and net class	LIMITED	✓
45-degree and memory pattern routing	LIMITED	✓
Interactive routing with shoving and plowing	LIMITED	✓
Interactive floorplanning	LIMITED	✓
Online design rule checking	LIMITED	✓
Flip, rotate, align, push, and move components	LIMITED	✓
Placement density analysis	LIMITED	✓
<b>Constraints Manager</b>		
Physical rules	LIMITED	✓
Spacing rules	LIMITED	✓
SameNet rules	LIMITED	✓
Properties and DRCs	LIMITED	✓
Differential pairs and static phase control	LIMITED	✓
Region rules	LIMITED	✓
Single line impedance rules	LIMITED	✓
Min. / Max. propagation rules	LIMITED	✓
Relative propagation rules	LIMITED	✓
Layer set rules	LIMITED	✓
Electrical constraint rule set (ECSets) / topology apply	LIMITED	✓
Matched group rules	LIMITED	✓
Total Etch Length	LIMITED	✓
<b>'SI' Signal Explorer</b>		
Pre-route signal integrity analysis	LIMITED	✓
Graphical topology definition and exploration	LIMITED	✓
Interactive waveform viewer	LIMITED	✓
OrCAD Capture SI integration and flow	LIMITED	✓
Macro modelling support (DML)	LIMITED	✓
IBIS 5.0 and ICM model support	LIMITED	✓
Spectre®-to-DML and HSpice-to-IBIS conversion	LIMITED	✓
Lossy transmission lines	LIMITED	✓
Coupled (3 net) simulation	LIMITED	✓
Differential-pair exploration and simulation	LIMITED	✓
Single net extraction from PCB Editor	LIMITED	✓
Post- route signal integrity analysis	LIMITED	✓