



# Power IC Model Library

## Improved simulation speed and accuracy

### About AEI Systems

The Power IC Model Library is a product of AEI Systems, a leader in the technology of SPICE and PSpice modeling, and Worst Case Circuit Analysis.

AEI Systems ([www.aeng.com](http://www.aeng.com)) specializes in the design and analysis of power conversion circuits for satellite systems, and also performs analysis for commercial and military, ground based and airborne, analog, digital and mixed mode applications.

EMA Design Automation provides exclusive distribution and support throughout North America for the AEI Systems Power IC Model Library.

### For More Information

For sales and pricing information contact EMA Design Automation, a Cadence Channel Partner.

EMA Design Automation, Inc.  
225 Tech Park Drive  
Rochester, New York 14623

Phone: 585.334.6001  
Fax: 585.334.6693  
eMail: [info@ema-eda.com](mailto:info@ema-eda.com)  
Web: [www.ema-eda.com](http://www.ema-eda.com)

AEI Systems® has developed proprietary relationships with the majority of the top analog IC manufacturers and is the sole developer of power IC models in many cases. These relationships ensure that the models exhibit the accuracy you expect from a PSpice simulation.

### Simulate your entire power system

The Power IC Model Library includes model netlists in PSpice syntax as well as symbols, model libraries and schematic sets for both Cadence OrCAD® Capture and Microsim Schematics. Also included are a set of example application circuits for many of the IC models.

The models utilize analog behavioral and special Boolean logic elements. Together they greatly reduce the model's simulation runtime, an important factor in SMPS analysis. Most simulations run in just a few minutes.

### The PSpice simulator provides speed increases for complex power supply simulations

Newly added models in the Power IC Model Library incorporate recent PSpice built-in component and building block advances. These advances allow models to take full advantage of PSpice's improved capabilities, which include speed increases for simulation of math equations and "if-then-else" constructs that are used throughout the model library. In addition, the models utilize analog behavioral elements and special Boolean logic elements. These improvements result in speed increases of 10% to 40% for power supply simulations, with most simulations capable of running in just a few minutes.

### The PSpice Model Library includes

- Over 200 Power IC Models (LIB files) including:
  - Phase shift, voltage and current mode PWM controllers, switching regulators, motor controllers, power factor correction, and power MOSFET Drivers
  - Popular parts: HS117/LM117, UC384x, UC152x, UCC380x, NCP1653, LT124x, UC182x, UC1846, TL431, IR2110, UC1854, TNY256, MIC44xx, UCC3895, TPS40055
  - Models for Texas Instruments, Intersil, ON Semiconductor, Linear Technology, Analog Devices, National Semiconductor, International Rectifier, Intersil, Micrel, Vishay and others
  - Linear ICs: AD813x Differential Amps, AD8333 Phase Shifter, AD8331 VGA, ADx36/x37 DC-RMS Converter
  - Nonlinear magnetic cores, transformers and opto-couplers
  - Power MOSFET drivers
  - New Spark Gap, Fluorescent Tube and Dead Time Controllable FET Driver Models
- Useful application circuit example schematics for most parts (DSN, OPJ, SCH files)
- Symbols for both OrCAD Capture and Microsim Schematics (OLB, SLB files) and support for symbol and schematic formats of both products

### System requirements

- OrCAD Capture version 9.2.3 or greater
- Cadence PSpice A/D version 9.2.3 or greater

©2008 EMA Design Automation, Inc. All rights reserved in the U.S. and other countries. EMA Design Automation and the EMA logo are registered trademarks of EMA Design Automation, Inc. Cadence, OrCAD, and PSpice are registered trademarks of Cadence Design Systems, Inc. Power IC Model Library and AEI are trademarks of AEI Systems, LLC. All other marks are the property of their respective owners.

