

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) 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: 877.362.3321
Fax: 585.334.6693
eMail: info@ema-eda.com
Web: www.ema-eda.com

©2021 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.

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 library includes example application circuits for most of the IC models. Symbols, model libraries and schematic sets are available for both Cadence OrCAD[®] Capture and Microsim Schematics. You can perform high-speed cycle-by-cycle simulation to show true large-signal performance, simulate current-mode control using the latest accurate modeling techniques, run CCM and DCM converter simulations, analyze control systems including loop gain, input filter design and analysis, and measure power stage loss and stress analysis for all major components. In summary, you can simulate your entire power system.

The PSpice simulator provides speed increases for complex power supply simulations

Some newly added models in the Power IC Model Library for PSpice incorporate recent PSpice built-in component and building block advances that allow elements. Together they greatly reduce the model's simulation runtime, an important factor in SMPS analysis. Most simulations run in just a few minutes. These improvements result in speed increases of 10 to 40% for power supply simulations.

System requirements

- OrCAD Capture/PSpice version 15.9 or greater
- Microsim Schematics/PSpice version 8.0

Key Benefits:

- Analyze large signal effects like start-up transients, power stage semiconductor stress, and step-load response
- Explore different approaches to transformer, converter, filter, and control structures
- Compute component stresses and test for excessive power dissipation
- Compare circuit characteristics with linear and nonlinear magnetics
- Analyze in both time and frequency domains
- Simulate and analyze your entire power supply without ANY limitations.

The PSpice Model Library Contains

- Over 1200 Power IC Models (LIB files) including:
 - o Phase shift, Voltage and current mode PWM controllers, Switching regulators, Motor controllers, Power factor correction, and Power MOSFET drivers
 - o Automotive Stimulus Models: ISO 76750/ISO 7637-2 and FMC1278 including C1220 (load dump), C1230A (power cycling), C1250 (voltage offset), and C1260A-E (Voltage dropout).
 - o Linear ICs: AD813x Differential Amps,
 - o Popular parts: UCC3895, HS117/LM117, UC384x, UC152x, UCC380x, LT124x, UC182x, UC1846, TL431, IR2110, UC1854
 - o Linear ICs: AD813x Differential Amps, AD8333 Phase Shifter, AD8331 VGA, ADx36/x37 DC-RMS Converter
 - o Nonlinear Magnetic Cores, Transformers, Opto-Couplers
 - o Texas Instruments, Intersil, ON Semiconductor, Linear Technology, International Rectifier, Micrel, Vishay and more
 - o 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 (OLB, SLB files)
- Support for both OrCAD Capture and Microsim Schematics symbol and schematic formats

