



FOR IMMEDIATE RELEASE

Contact Information:

EMA Design Automation
Janet Roberts
949-443-1695
Janet@GJRoberts.com

AEi Systems
Lee Weinberg
310-216-1144
Lee@AENG.com

**Power IC Model Library for Cadence PSpice, Ups LED and
LED Driver Plus TI Listings in New Release 4.0;
Now over 400 Models**

Rochester, NY (October 10, 2011) – EMA Design Automation™ (www.ema-eda.com), a full-service provider of Electronic Design Automation (EDA) solutions, and AEi Systems, a world leader in power electronics modeling, analysis and design, today announced the release of version 4.0 of AEi Systems' Power IC Model Library for the Cadence® PSpice® simulator.

Version 4.0 has over 400 time-domain simulation models for power electronic designs. Several previously unavailable Texas Instruments (TI) models debut in version 4.0, and future Library updates are anticipated to focus on many of TI's newest and most popular components.

The Power IC Model Library includes model netlists in PSpice syntax, schematic symbols for both Cadence OrCAD® Capture and legacy schematics, and a set of example application circuits for many of the IC models. The models are compatible with OrCAD Capture version 16.x software.

The Library is available for purchase. AEi Systems and EMA are keeping the price of version 4.0 of the Library the same as the current version: \$1,995, plus yearly maintenance of \$495.

Important breakthroughs for power engineers

“The Power IC Model Library for PSpice has models of parts that are simply not available from any other EDA company,” said Manny Marcano, President and CEO of EMA Design Automation. “AEi Systems has proprietary relationships with nearly all of the top analog IC manufacturers -- relationships that provide unique access to the part characteristics needed to produce models with the accuracy our customers expect. At \$2,490 including the first year’s maintenance, this library is a tremendous value; each model costs from \$2,000 to \$15,000 to develop.”

“This library makes the PSpice simulator even more compelling for engineers in the power supply market. Many EDA vendors only have access to information in the manufacturer data sheets. This is simply not sufficient to create a precise model of a controller or regulator,” stated Charles Hymowitz, Managing Director of AEi Systems. “Data sheets do not have the level of detail required, so those companies and modelers who rely solely on data sheet input will necessarily produce inferior, inaccurate models.”

New additions for version 4.0 of the Power IC Model Library include:

- TPS2490/91, TPS40200, TPS51217, TPS51220A, TPS51427/A,
- TPS53311, TPS54218, TPS54225/6, TPS54240/60, TPS54311-16,
- TPS54319, TPS54325/6, TPS54425, TPS54611-16, TPS61040/41,
- TPS62060, TPS62065, TPS62067, TPS62120/22, TPS62230/31,
- TPS622310-14, TPS62232-39, TPS62650
- Everlight Opto-Couplers, 6N136/137
- LED Models

New Application Examples Include:

- Switch Boost Converter in SOT-23 for LCD and White LED Applications
- High-Voltage Power-Limiting Hot Swap Controller
- Non-Synchronous Buck DC/DC Controllers

- Synchronous Step Down Controllers for Notebook Power Rails
- Current-Mode Notebook System Pwr Controller

Some of the Power IC Model Library's newly-added models incorporate recent PSpice built-in component and building block advances. These advances allow models to take full advantage of new PSpice capabilities including increasing speed for simulating math equations and "if-then-else" constructs that are used throughout the model library. This improvement results in speed increases of 10 to 40% for power-supply simulations.

Complex models, verified with bench data

SMPS applications are more demanding than ever. Today's designs require increases in power IC functionality, switching frequency, and system interaction. State space based models simply do not reveal many important nonlinear factors that influence these performance characteristics.

The models in the Power IC Model Library enable designers to perform high-speed, cycle-by-cycle simulations to show true large-signal performance, simulate current-mode control using the latest accurate modeling techniques, run CCM and DCM converter simulations, generate line and load step responses, and measure power stage loss and stress for all major components.

The models in the Power IC Model Library are compared and verified with bench data under startup, steady state, line, and load transient conditions. Nonlinear characteristics such as propagation delay, switching speed, drive capability, maximum duty cycle/current limits, and startup phenomena are all accurately modeled.

Designers can directly compare the performance of components from various vendors and analyze the effects of different implementations such as peak current mode control, hysteretic current control, low voltage, and low operating current, to name just a few.

The Power IC Model Library for PSpice also includes:

- Popular parts: UCC3895, HS117/LM117, UC384x, UC152x, UCC380x, LT124x, UC182x, UC1846, TL431, IR2110
- Nonlinear Magnetic Cores, Transformers, Opto-Couplers
- Models for parts from Texas Instruments, Intersil, ON Semiconductor, Linear Technology, International Rectifier, Micrel, Vishay, and others

More details are available at www.ema-eda.com/products/other/powericlib.aspx, including a list of the components and documentation detailing the models' accuracy and performance.

About AEi Systems

AEi Systems is a world leader in SPICE modeling, worst case, failure and reliability analysis, and power systems analysis and design. AEi Systems serves nearly every significant IC and aerospace manufacturer and many of their customers. AEi Systems' headquarters are in Los Angeles, California. For more information on AEi Systems, please contact the company at 310-216-1144 or visit the AEi Systems website at www.AENG.com.

About EMA Design Automation, Inc.

EMA Design Automation is a full-service provider of Electronic Design Automation (EDA) solutions including a complete range of software tools, consulting services, product training, and technical support for the entire PCB and custom integrated circuit design process. EMA is a Cadence® Channel Partner serving all of North America and has been a Cadence distributor since 1998. EMA manufactures TimingDesigner, a static timing analysis solution, and distributes it globally through a worldwide network of value added resellers. EMA is a privately held corporation headquartered in Rochester, New York. Visit EMA at www.ema-eda.com for more information.

EMA Design Automation is a trademark and TimingDesigner is a registered trademark of EMA Design Automation, Inc. Cadence, OrCAD, and PSpice are registered trademarks of Cadence Design Systems, Inc. All other logos and trademarks in this release are the property of their respective owners.

###