



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 Expanded in New Release

Rochester, NY (September 20, 2007) – 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 and analysis, today announced the release of version 2.0b of AEi Systems' Power IC Model Library for the Cadence® PSpice® simulator.

Version 2.0b has over 200 time-domain simulation models for power electronic designs. Several previously unavailable Texas Instruments (TI) models debut in version 2.0b, and future 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 all past versions of the PSpice simulator, as well as the latest OrCAD Capture version 9.x, 10.x, and 16.0 software. The price of version 2.0b of the library is \$1,995, plus yearly maintenance of \$495.00.

Important breakthroughs for power engineers

“The Power IC Model Library for PSpice contains 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, the library is a tremendous value, as single models cost from \$2,000 to \$15,000 and more apiece to develop.”

“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 simply do not have the level of detail required, so those companies who rely solely on data sheet input will produce substandard and inaccurate models.”

“The Power IC Model Library is an important addition to a PSpice simulation environment. This library makes the PSpice simulator even more compelling for engineers in the power supply market. EMA has a skilled team of SPICE experts, making them the perfect partner for us,” stated Hymowitz.

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 analysis 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, hysteric current control, low voltage, and low operating current, to name just a few.

New additions to the Power IC Model Library include:

- Power MOSFET Drivers: SIC7x0DD, MIC44xx series
- Linear parts: AD813x Differential Amplifiers, AD833x VGAs, ADx36/x37 RMS-DC Converters

- Popular Parts: HS117R, LM117, UCC380x Current-Mode PWM, NCP1653 PFC, MC34063/MC33363 Regulators, TPS40055, UCC380x, and UCC3809 Controllers
- Generic Saturable Magnetic Core models, Transformers, Opto-Couplers
- Spark Gap Model
- Fluorescent Tube Model

The Power IC Model Library for PSpice also includes:

- Phase shift, voltage and current mode PWM controllers, Switching regulators, Motor controllers, Power factor correction, Power MOSFET Drivers
- Popular parts: UC384x, UC152x, HIP2100, LT124x, UC182x, UC1846, TL431, IR2110, 57xxx, UC1854, TNY256, MIC44xx,
- 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 trademarks in this release are the property of their respective owners.

###