ISSUE: January 2024

SPICE Library Adds SiC Power Devices And IGBTs

ROHM Semiconductor has expanded its SPICE model lineup for the LTspice circuit simulator, increasing its number of models to more than 3,500 for discretes (which can be downloaded from product pages). In addition to the existing lineup of bipolar transistors, diodes, and MOSFETs, ROHM has added SiC power devices and IGBTs to the library (see the figure). This brings the coverage of LTspice models on ROHM's website to over 80% of all products, providing greater convenience to designers when using circuit simulators that incorporate discrete products, now including power devices.

In addition, ROHM has added a Design Models page to their website that allows simulation models to be downloaded directly. Documentation on how to add libraries and create symbols (schematic symbols) is also available to facilitate circuit design and simulation.

In recent years, the increasing use of circuit simulation for circuit design has expanded the number of tools being utilized. Among these, the free LTspice program is an attractive option for a range of users, from students to seasoned engineers at well-known companies.

Going forward, ROHM will continue to contribute to solving circuit design issues by expanding the number of models compatible with various simulators, while providing web tools such as ROHM Solution Simulator to meet growing customer needs.

A variety of simulation models are offered that support LTspice and other tool environments. For more information see the Design Models page and the ROHM Solution Simulator page. Also see the application notes "How to Use LTspice Models" and "How to Use LTspice Models: Tips for Improving Convergence".
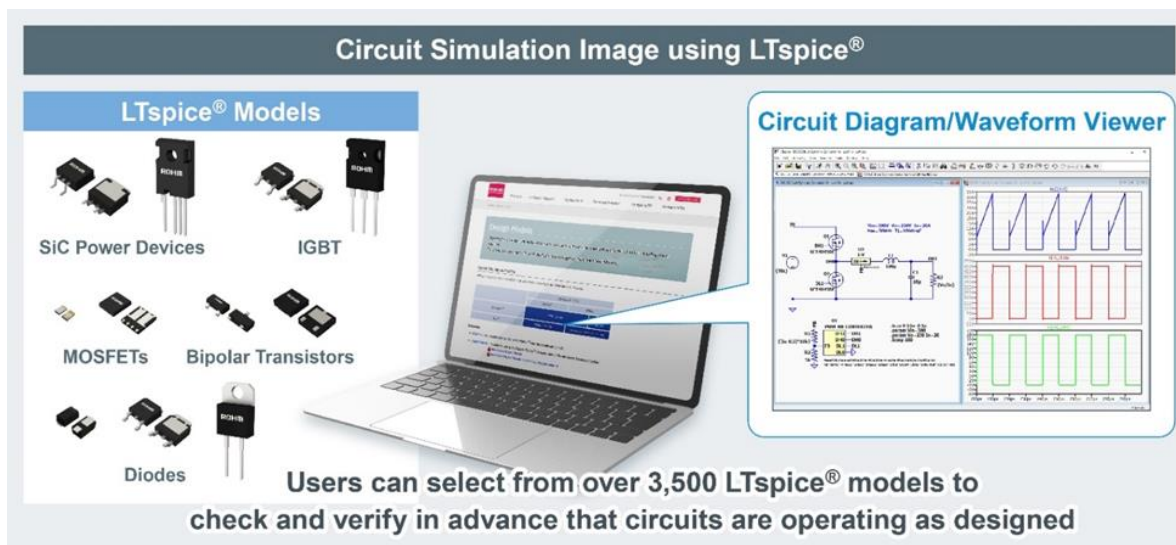

Figure. ROHM Semiconductor has expanded its SPICE model lineup for the LTspice circuit simulator, increasing its number of models to more than 3,500 for discretes. This brings the coverage of LTspice models on ROHM's website to over 80% of all products.