Energy Efficiency Through Innovation

APPLICATION NOTE
AN1003

SiC Schottky MPS™ Diodes
SPICE Model Usage Instructions

CONTENTS

This application note provides information on installing and using GeneSiC Semiconductor's SPICE models. The SPICE model package for each part consists of:

1) Encrypted SPICE model library (.lib)
2) Schematic symbols (.asy)
3) Disclaimer

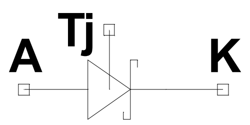
The following points should be noted by the user while using these SPICE models:

- The models are designed to be accurate over operating temperature range specified on the product datasheet
- In case of dual diode (SOT-227) and common cathode packages (TO-247-3), the models are per leg.

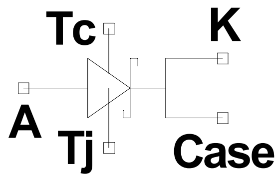
1 – Model Description

The Level-III model, often known as the self-heating electro-thermal model, includes the values of the internal thermal heat dissipation network to estimate the device's operating junction temperature. T_C represents the case temperature and the user can set its value by connecting a voltage source referenced to the ground. The voltage measured at T_j estimates the internal junction temperature of the device and it should be noted that the absolute maximum temperature at T_j should not exceed 175 °C under normal operating conditions. The remaining ports – Anode (A), Cathode (K) and Case (electrically connected to Cathode for non-isolated base-plate / case type packages), are connected into the electrical circuit. A heat-sink may be included / modeled as an external RC network to the node T_C.

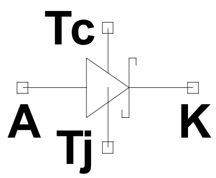
GeneSiC's LTSPICE schematic symbol files are categorized by the discrete / module device package type as listed below –



DO-214 (SMB)
TO-220FP



TO-252-2
TO-263-7
TO-220-2
TO-247-2
TO-247-3



SOT-227
(Isolated Base-plate)

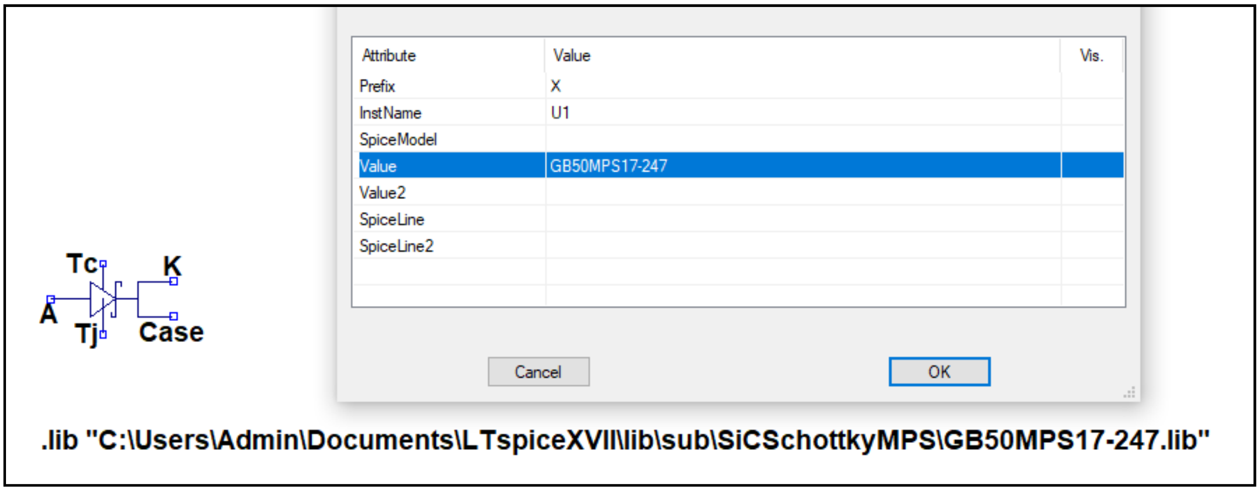
3 – Installation

These instructions are for LTSPICE users and provide guidance on installing GeneSiC Semiconductor's SPICE symbols and model libraries on the user's system.

1. After unzipping the package downloaded from the website, copy the contents (.asy files) of the "Schematic Symbols" folder containing SPICE symbols (.asy files) into your default LTSPICE symbols library directory.
   (Example Location: ***C:\Users\Admin\Documents\LTspiceXVII\lib\sym\SiCSchottkyMPS***)
2. Place the specific device model library file (.lib files) into your default LTSPICE directory / sub-circuit library folder. Remember this path as it will be needed to enter into LTSPICE later.
   (Example Location: ***C:\Users\Admin\Documents\LTspiceXVII\lib\sub\SiCSchottkyMPS***)
3. Open or restart LTSPICE to load the new symbols.

4 – Usage

1. Place the GeneSiC package symbol (Example: **TO-247-2_SiCSchottkyMPS.asy**) into the LTSPICE schematic using the Edit>Component selection dialog box.
2. Add a .lib SPICE directive which will link the relevant GeneSiC model library file to this schematic. Click Edit>SpiceDirective and a text box will appear.
   Enter **.lib *"C:\Users\Admin\Documents\GeneSiC\GB50MPS17-247.lib"***
3. Right Click on the GeneSiC component and edit the Value attribute, as shown in the figure below. Update the value of this attribute to GeneSiC part name (Example: ***GB10MPS17-247***)



Disclaimer

Models provided by GeneSiC Semiconductor are not warranted as fully representing all of the specifications and operating characteristics of the product to which the model relates. The model describes the characteristics of a typical product, and in all cases, the most recent revision of the product's datasheet is the final design guideline and the only actual performance specification. Although SPICE models can be a useful tool in evaluating device performance, they cannot model the exact device performance under all conditions, nor are they intended to replace the design verification process. Therefore, GeneSiC Semiconductor does not assume any liability arising from their use and reserves the right to change models without prior notice.