Wolfspeed E-Series SiC Schottky Diodes: SPICE Model Usage Instructions

Thank you for using Cree SiC Schottky diodes. Our diode Spice model package includes these files:
- *Wolfspeed E4D E-Series Packaged SPICE Model Library.lib* – SPICE models for the E4D family of discrete packaged diodes
- *Wolfspeed EPW4 E-Series Bare Die SPICE Model Library.lib* – SPICE models for the EPW4 family of bare die diodes
- *E-Series_LTSPICE_Symbols.zip* – diode symbols for LTspice users

Notes:
- The model is designed to be accurate over the ranges presented in the corresponding datasheet typical characteristic curves.
- The model is valid over the 25˚C to 175˚C temperature range.
- Reverse leakage currents are modeled as a breakdown voltage. No low current detail is provided.
- The surge response of the PiN structure is not included in this version.
- Recombination current is assumed to be zero.
- Currently, the model is electrical only. No thermal model is included.
- LT Spice users: please note that a LTspice symbol library is included in this model kit. Specialized instructions for the installation and usage of these symbols are located later in this document.

Installation Instructions (Standard users):
1. Place "Wolfspeed E4D E-Series Packaged Spice Model Library.lib" and "Wolfspeed EPW4 E-Series Bare Die Spice Model Library.lib" into a directory of the user's choice.

Usage Instructions (Standard users):
1. Start SPICE.
2. Create a new schematic/netlist (or open an existing one).
3. Add the .lib SPICE directives which will link the relevant Cree library files to this schematic/netlist. Point the directives to the appropriate directory <path> where the two .lib files were unzipped during installation.

Installation Instructions (LTspice users):
1. Place "Wolfspeed E4D E-Series Packaged Spice Model Library.lib" and "Wolfspeed EPW4 E-Series Bare Die Spice Model Library.lib" into a directory of the user's choice. Remember this path as it will be need to be entered into LTspice later.
2. Locate the LTspice symbols library directory. By default on a Windows 7 64-bit PC, this directory is located at "C:\Program Files (x86)\LTC\LTspiceIV\lib\sym"

3.   Unzip "E-Series_LTSPICE_Symbols.zip" into this LTspice symbols library directory. A new subdirectory named "ESeriesDiodes" should be automatically created here, and this subdirectory should contain a number of .asy files. If the subdirectory "ESeriesDiodes" already exists, the .asy files will be unzipped to this folder. Each file corresponds to a particular E-Series diode part number.

4.   Open or re-open LTspice to load the new symbols.

Usage Instructions (LTspice users):

1.   Start LTspice.
2.   Create a new LTspice schematic (or open an existing one).
3.   Add a .lib spice directive which will link the relevant Cree library files to this schematic. Click the Edit menu, then click Spice Directive. A textbox will appear. Enter the text ".lib C:\<path>\ Wolfspeed E4D E-Series Packaged Spice Model Library.lib" without the quotes, and insert the appropriate directory <path> where the two .lib files were unzipped during installation.
4.   Click OK. Position the directive textbox somewhere on the schematic, then click to place it.
5.   Repeat steps 3-4 and add a second directive ".lib C:\<path>\Wolfspeed EPW4 E-Series Bare Die Spice Model Library.lib" to the schematic.
6.   Click the Edit menu, then click Component. The component selection dialog box will appear. Double-click the [ESeriesDiodes] group in order to show the Cree Diode symbols. Click the part number of interest and then click OK.
7.   Place the symbol and wire it up as normal for a diode. Note that an electrical terminal corresponding to the case / back metal tab of the package is also available for use in the schematic. The case terminal can be left floating if not used in the application.

tjb
July 27 2018