User Guide

Wolfspeed SiC Schottky Diode LTspice Model

Quick Start Guide

Contents

1. Introduction....2
2. LTspice Software....2
   2.1 Prerequisite:....2
   2.2 Package Contents:....2
   2.3 Software Requirement:....2
   2.4 Model Installation Guidelines:....3
3. Model Specifications....3
   3.1 Model features....3
4. Simulation Guidelines:....3
5. Migrating Wolfspeed LTspice model to others SPICE software....6
6. Revision History....6

© 2024 Wolfspeed, Inc. All rights reserved. Wolfspeed® and the Wolfstreak logo are registered trademarks and the Wolfspeed logo is a trademark of Wolfspeed, Inc. Other trademarks, products, and company names are the property of their respective owners and do not imply specific product and/or vendor endorsement, sponsorship, or association. This document is provided for informational purposes only and is not a warranty or a specification. For product specifications, please see the data sheets at www.wolfspeed.com.

DISCLAIMER

Models provided by WOLFSPEED are not warranted by WOLFSPEED as fully representing all the specifications and operating characteristics of the semiconductor product to which the model relates. The model describes the characteristics of a typical device. In all cases, the current data sheet information for a given device is the final design guideline and the only actual performance specification. Although models can be a useful tool in evaluating device performance, they cannot model exact device performance under all conditions, nor are they intended to replace laboratory testing for final verification. This model is preliminary and subject to change without notice. Wolfspeed will not be responsible for any error or simulation issue arising due to the editing of the model library file.

1. Introduction

The primary intention of building a LTspice model is to allow the users to create a converter circuit and understand its design and performance parameters. With the simulation of LTspice model, designers can save a lot of time by reducing the design cycles that lead to the early introduction of the product to the market. Wolfspeed Schottky diode LTspice models contain either Level 1 (constant temperature) model or Level 3 (electro-thermal) model. Level 1 model considers the device has a constant junction temperature (temperature is equal to the environment temperature) during the transient simulation. This model is less complex than level 3 thus it takes less computing time for a simulation. Level 3 models have Tc & Tj terminals included as a provision of self-heating to observe the change in junction temperature and performance of the device. Level 3 model integrated with PiN structure for surge response. These models are a useful tool in evaluating device performance during design stage, but the model cannot perform exactly under all conditions, nor are they intended to replace prototype.

2. LTspice Software

2.1 Prerequisite:

LTspice simulation software (https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html)

2.2 Package Contents:

- Level 1 SPICE Library Packaged Device Model (Wolfspeed Power CXD Packaged SPICE Model Library.lib) – Library consists of same generation and series Schottky diodes.
- Level 3 SPICE Library Packaged Device Model (CXD0XXXXXXX.lib) – Each diode has its own library that name after the part number.
- LTSPICE Device & Die Symbol (Part_number.asy)

2.3 Software Requirement:

This model has been developed and optimized for LTspice. It is the responsibility of the user to be well-versed with the basic operation of LTspice simulation tool. Using this model library on other SPICE simulation tool may result in convergence error or incorrect simulation result. Please use the recommended software or verify the result before use.

© 2024 Wolfspeed, Inc. All rights reserved. Wolfspeed® and the Wolfstreak logo are registered trademarks and the Wolfspeed logo is a trademark of Wolfspeed, Inc. Other trademarks, products, and company names are the property of their respective owners and do not imply specific product and/or vendor endorsement, sponsorship, or association. This document is provided for informational purposes only and is not a warranty or a specification. For product specifications, please see the data sheets at www.wolfspeed.com.

2.4 Model Installation Guidelines:

1. Download the LTspice model at http://go.wolfspeed.com/all-models

2. Extract the zip file.

3. Verify the presence of all the files indicated in the package contents.

4. Copy the Wolfspeed device symbol file (.asy) and paste it into the LTspice symbol directory. Typical installation path is given by (C:\Program Files (x86)\LTC\LTspiceIV\lib\sym). It is recommended to create a folder just for Wolfspeed Schottky diode at the path mentioned above. This would make the device symbol appear in the component selection window. A software restart may be required to observe the change.

5. The device symbol will be like the one shown in figure 1.

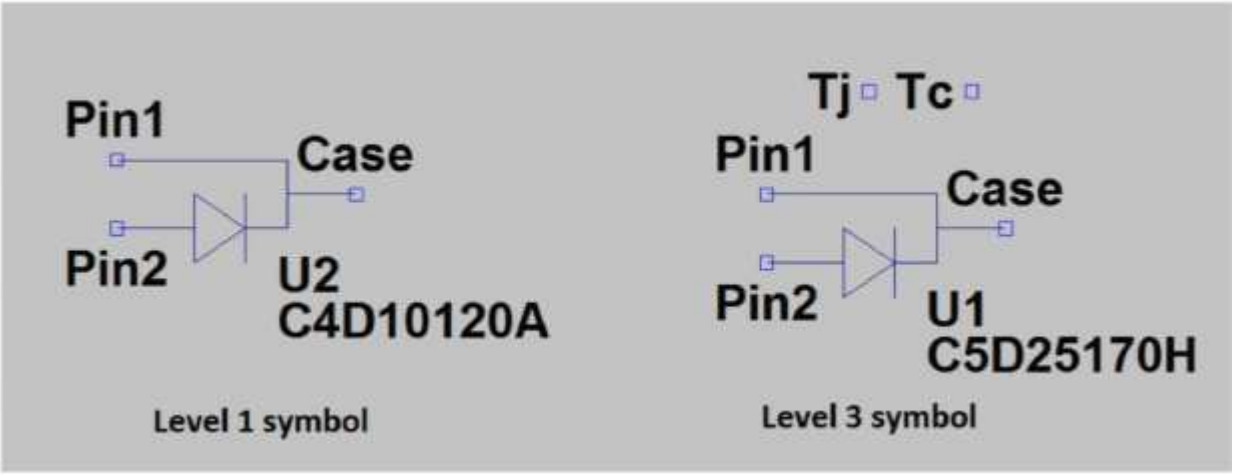


Figure 1: Device symbol in a schematic (reference purpose only).

6. Copy the Wolfspeed Schottky diode library file (.lib) and paste it into the LTspice library directory. Typical installation path is given by (C:\Program Files(x86)\LTC\LTspiceIV\lib\).

3. Model Specifications

3.1 Model features

- Optimized for -55°C & 175°C temperature.
- Valid for temperature range -55°C to 175°C
- Level 3 model includes self-heating and transient thermal capability.
- Level 3 model has integrated with PiN structure for surge response.
- Parasitic inductance associated with electrodes will be included in the model.
- Reverse leakage currents are modeled as a breakdown voltage. No low current detail is provided.

4. Simulation Guidelines:

The SiC LTspice model is provided with the following terminals:

- Anode

© 2024 Wolfspeed, Inc. All rights reserved. Wolfspeed® and the Wolfstreak logo are registered trademarks and the Wolfspeed logo is a trademark of Wolfspeed, Inc. Other trademarks, products, and company names are the property of their respective owners and do not imply specific product and/or vendor endorsement, sponsorship, or association. This document is provided for informational purposes only and is not a warranty or a specification. For product specifications, please see the data sheets at www.wolfspeed.com.

- Cathode
- Case
- Junction Temperature terminal - Tj
- Case Temperature – Tc (Except on die models)

Each diode has its own dedicated symbol, for example C4D10120A Schottky diode has a symbol C4D10120A.asy . Users are required to manually include the library path at each circuit design. This is to provide LTspice the path where the library is located. Example:

.lib "C:\Program Files (x86)\LTC\LTspiceIV\lib\C5D25170H.lib"



The model library file (.lib) should not be edited under any circumstance as it may result in convergence error, incorrect simulation result or longer simulation time.

The terminals Tj and Tc were specifically included in the design to analyze the self-heating of the device as a function of time. The terminal Tc represents the case temperature and Tj represents the junction temperature. The temperature connections are working as voltage pins. Therefore, a potential difference of 1V refers to a temperature difference of 1°C.



The Junction Temperature terminal (Tj) can either be used to read junction temperature or to apply a junction temperature. This terminal can be left floating.

The voltage at the Tj node contains the information about the time-dependent junction temperature which in turn acts directly on the temperature-dependent electrical model.



The Case Temperature terminal (Tc) must be connected to either a voltage source or a Heat Sink RC Network. This terminal can be left floating.

The Tc terminal should be connected to either a voltage source (which denotes the case temperature) or to an external RC network (heat sink model) to observe its effect on the junction temperature. Figure 2 shows the connection of Tc terminal to an ambient temperature of 25°C where figure 3 shows connecting external thermal network to Tc.

© 2024 Wolfspeed, Inc. All rights reserved. Wolfspeed® and the Wolfstreak logo are registered trademarks and the Wolfspeed logo is a trademark of Wolfspeed, Inc. Other trademarks, products, and company names are the property of their respective owners and do not imply specific product and/or vendor endorsement, sponsorship, or association. This document is provided for informational purposes only and is not a warranty or a specification. For product specifications, please see the data sheets at www.wolfspeed.com.

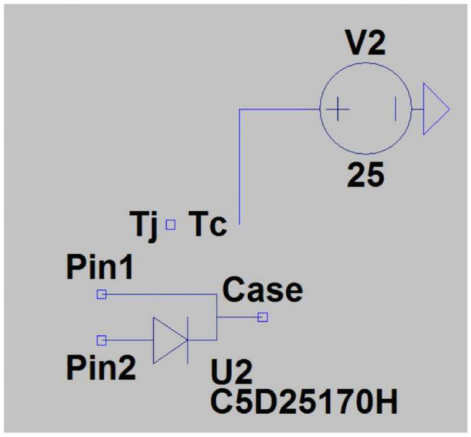


Figure 2: Fixing Case Temperature to 25°C

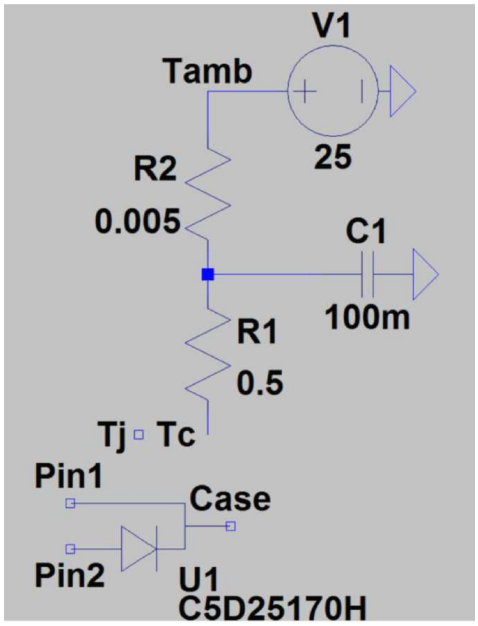


Figure 3: Connecting external thermal network to Tc

Either Tj or Tc must be connected to a voltage source to converge properly.

To perform DC simulation, the junction temperature (Tj) must be connected to a voltage source to fix the junction temperature to a constant value.

To use the model for generating DC characteristics at a junction temperature, the junction temperature must be fixed at a constant value. This can be achieved by connecting the terminal Tj to a fixed voltage source as shown in Figure 4.

© 2024 Wolfspeed, Inc. All rights reserved. Wolfspeed® and the Wolfstreak logo are registered trademarks and the Wolfspeed logo is a trademark of Wolfspeed, Inc. Other trademarks, products, and company names are the property of their respective owners and do not imply specific product and/or vendor endorsement, sponsorship, or association. This document is provided for informational purposes only and is not a warranty or a specification. For product specifications, please see the data sheets at www.wolfspeed.com.

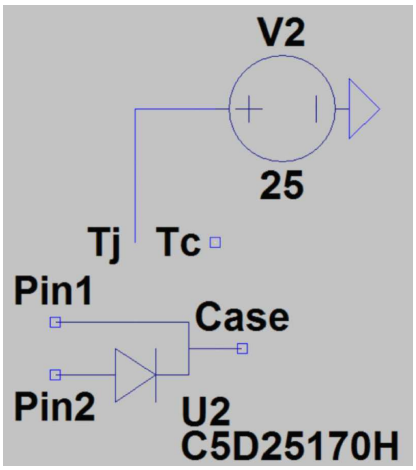


Figure 4: Junction maintained at a constant temperature of 25°C for DC simulation.

5. Migrating Wolfspeed LTspice model to others SPICE software

Wolfspeed Schottky diode SPICE models are both LTspice and OrCad Pspice compatible. To use Wolfspeed MOSFET SPICE model on other SPICE software, user may need to do few more steps to get it works. Some SPICE software use different extension like SIMetrix uses library with extension .lb whereas Ltspice and Pspice use extension .lib. Besides that, Pspice, Ltspice & SIMetrix use their own symbol format thus user should create their own symbol. Note: It is the responsibility of the user to verify the model against datasheet after changing the format of the model.

6. Revision History

Date Revision Changes
06/10/2019 V1.0 Initial release

Date	Revision	Changes
June 2019	1	Initial Release
January 2024	2	Rebranding

© 2024 Wolfspeed, Inc. All rights reserved. Wolfspeed® and the Wolfstreak logo are registered trademarks and the Wolfspeed logo is a trademark of Wolfspeed, Inc. Other trademarks, products, and company names are the property of their respective owners and do not imply specific product and/or vendor endorsement, sponsorship, or association. This document is provided for informational purposes only and is not a warranty or a specification. For product specifications, please see the data sheets at www.wolfspeed.com.