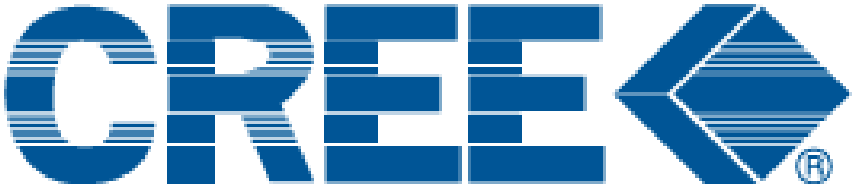

CREE C2M SiC MOSFET PSPICE MODEL

Quick start guide

REV 1.2

This document is prepared as a quick reference guide to perform simulations based on CREE SiC power MOSFET PSPICE library using LTSPICE simulation software. This user guide is applicable to the family of Generation II SiC MOSFETs.

DISCLAIMER

Models provided by CREE are not warranted by CREE as fully representing all of 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.

About C2M Spice Library Beta Version:

This package contains the beta version of CREE MOSFET C2M0XXX120D electro-thermal SPICE models for packaged device and bare die. The C2M family of SiC MOSFETs provides customers a wide range of products that includes all of the latest innovations in SiC developed by Cree. These modules replace the CMFXX models which Cree offered for the 1st generation of devices. The new models include self heating to observe the change in the device junction temperature. The model provides a reasonable approximation for the MOSFET in the third quadrant. However, the body diode threshold voltage was modeled with V_{GS} = -5V and assumes this is fixed for all values of V_{GS}. However, the effect of slight change in the threshold voltage of the body diode over the range of -5V ≤ V_{GS} ≤ 0V is not modeled. The Model is most accurate at the I_{DS} (DC) @ T_c=100°C rating operating conditions as shown in the device data sheet.

Prerequisite:

LTSPICE simulation software (http://www.linear.com/designtools/software/#LTspice)

Package Contents:

- SPICE Library Packaged Device Model (C2M0XXX120D - Packaged Beta.lib) – Model includes the TO-247 package parasitic.
- SPICE Library Bare Die Model (C2M0XXX120D - Die Beta.lib) – Model does not include any package parasitic.
- LTSPICE Device Symbol (power_nmos_heat.asy)

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 PSPICE simulation tool may result in convergence error or incorrect simulation result. Please use the recommended software.

Model Installation Guidelines:

1. Extract the zip file.
2. Verify the presence of all the files indicated in the package contents.
3. Copy the LTSPICE Device Symbol file and paste it into the LTSPICE symbol directory. Typical installation path is given by (C:\Program Files (x86)\LTC\LTspiceIV\lib\sym). This would make the device symbol appear in the component selection window. A software restart may be required to observe the change.
4. The Device symbol will be similar to the one shown in figure 1. LTSPICE provides the option for changing the visibility of the labels associated with the terminals.

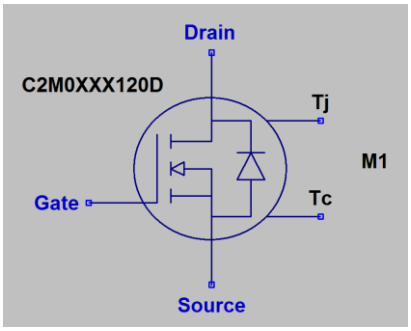

Figure 1. Device symbol in a schematic (reference purpose only).

5. The user can keep the library file at any location as long as the path is mentioned correctly. It is recommended to place the library file in the same folder used for simulation. Please refer to the LTSPICE manual for associating model library to a device symbol.

Simulation Guidelines:

The SiC electro-thermal PSPICE model is provided with the following terminals:

- Drain
- Gate
- Source
- **Tj**
- **Tc**

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 fix 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 should not be left floating.

The Tc terminal has to be connected to either a voltage source (which denotes the ambient 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.

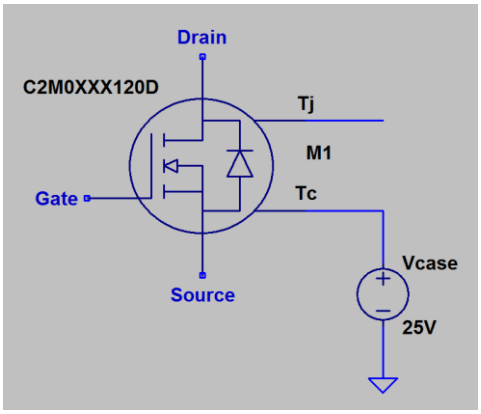

Figure 2. Fixing Case Temperature to 25°C

In order to perform DC simulation, the junction temperature (Tj) must be connected to a voltage source in order to fix the junction temperature to a constant value. The case temperature terminal can be set to 25°C.

In order to use the model for generating DC characteristics at a particular junction temperature, the junction temperature has to be fixed at a constant value. This can be achieved by connecting the terminal Tj to a fixed voltage source as shown in figure 3. The case temperature terminal can be connected to a 25V source.

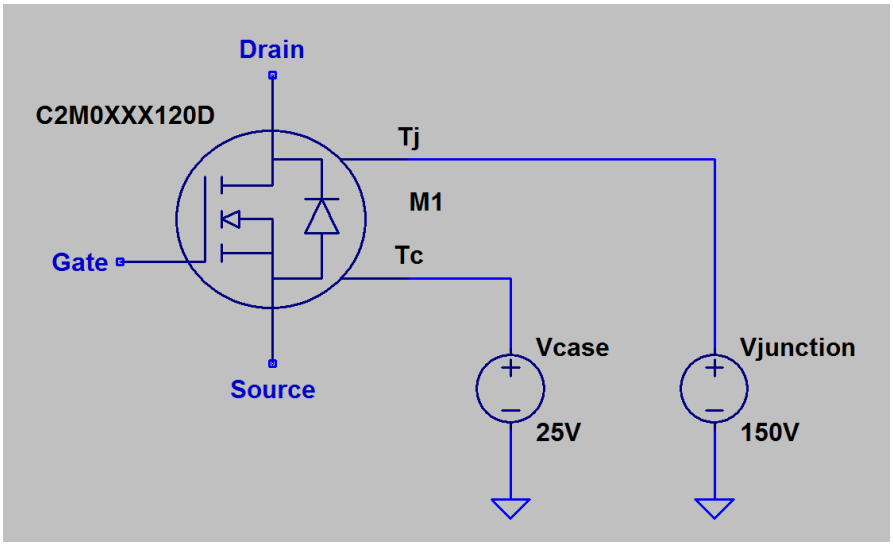

Figure 3. Junction maintained at a constant temperature of 150°C for DC simulation.

While performing simulations, it is recommended to change simulation settings by including the following SPICE directive in order to optimize the simulation speed and avoid any convergence error.

.OPTIONS METHOD=GEAR ABSTOL=1e-6 CHGTOL=1e-12 GMIN=1e-9 ITL1=1000 ITL2=1000 ITL4=1000 ITL6=1000 RELTOL= 0.001 VNTOL=1e-3 NOOPITER

Some of the settings in the software can also be modified for faster simulations. The following screenshot in fig. 4 shows the recommended settings. These settings can be accessed by going to: Simulate -> Control Panel -> SPICE.

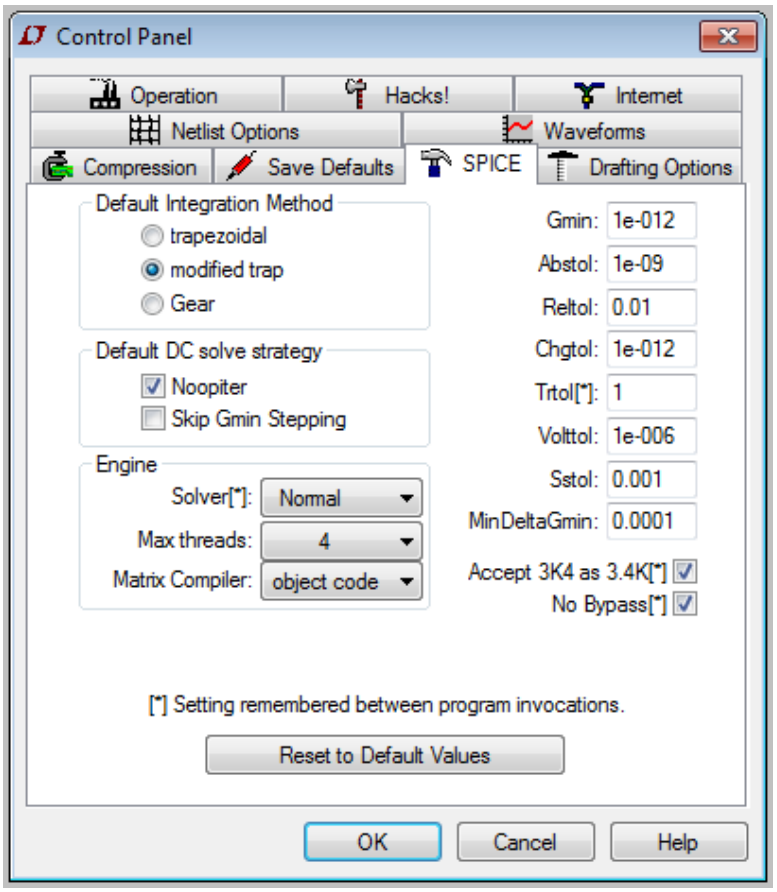

Figure 4. LTSPICE Simulation Settings

Simulation Example:

The schematic in figure 5 shows a clamped inductive switching circuit implemented in LTSPICE. The inductor is replaced by a constant current source for simplification. Various circuit parasitic components have been added to better match the experimental setup and get more accurate simulation results.

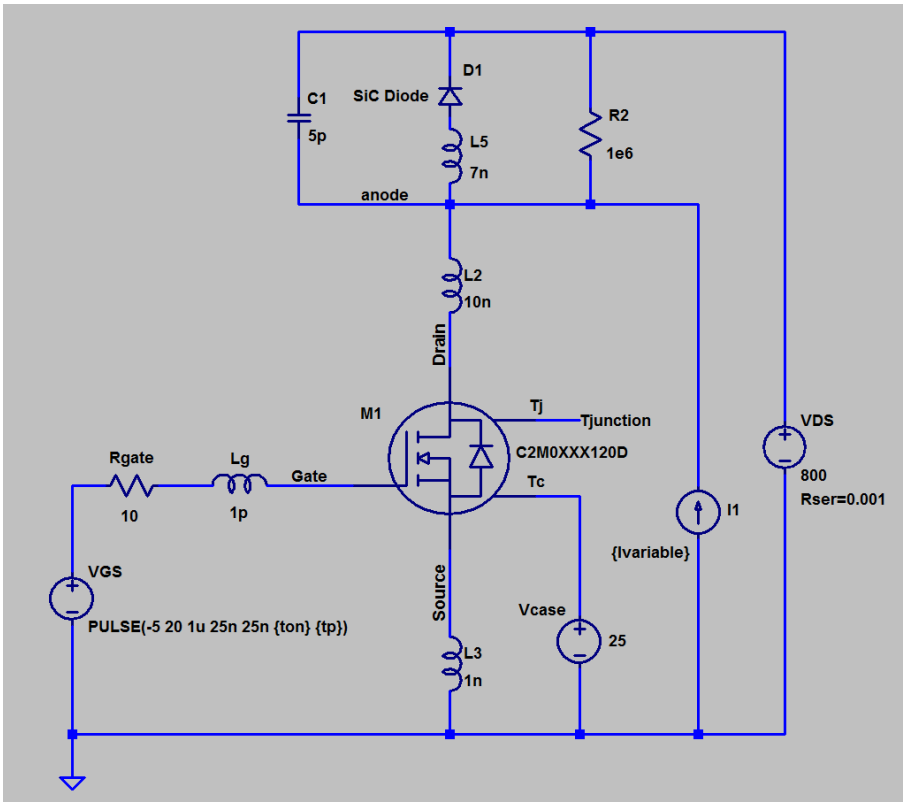

Figure 5. Clamped Inductive Switching Circuit

The Drain Voltage (Blue) and Drain Current (Red) waveforms are shown in figure 6.

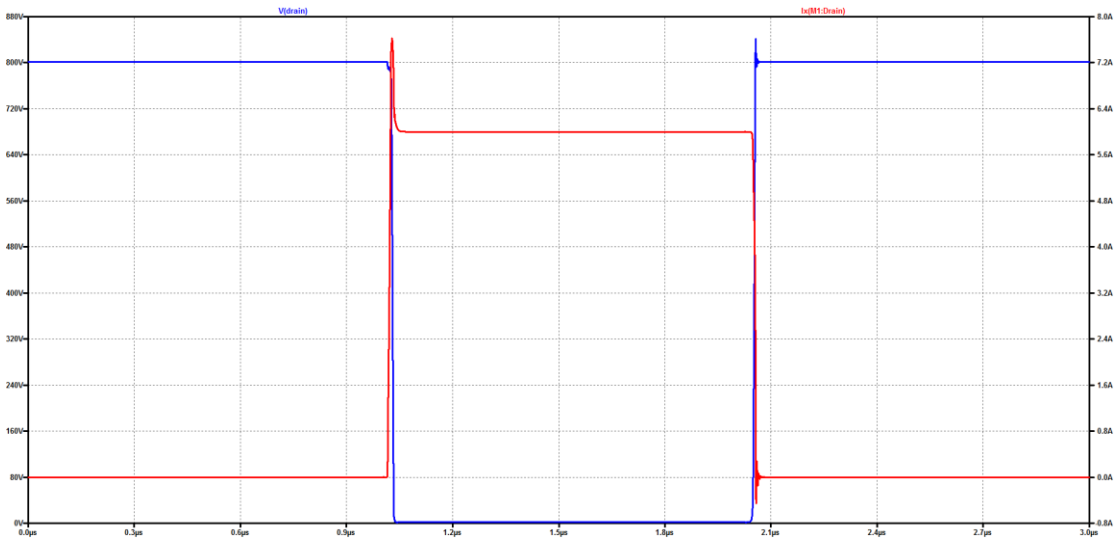

Figure 6. Waveform Screenshot obtained from LTSPICE Simulation

APPENDIX

Specific Simulation Requirements

1.	**Part No: C2M0080120D**	**R_{DSON}: 80mΩ**

Simulation of simple circuits do not require the altering of simulation settings however, while simulating complex circuits, it is highly recommended to include the SPICE directive (specified in the document) to avoid convergence error and speed up the simulation.

2.	**Part No: C2M0280120D**	**R_{DSON}: 280mΩ**

It is *mandatory* to include the SPICE directive to avoid convergence error and speed up the simulation process.

3.	**Part No: C2M0025120D**	**R_{DSON}: 25mΩ**

It is highly recommended to include the SPICE directive (specified in the document) to speed up the simulation.