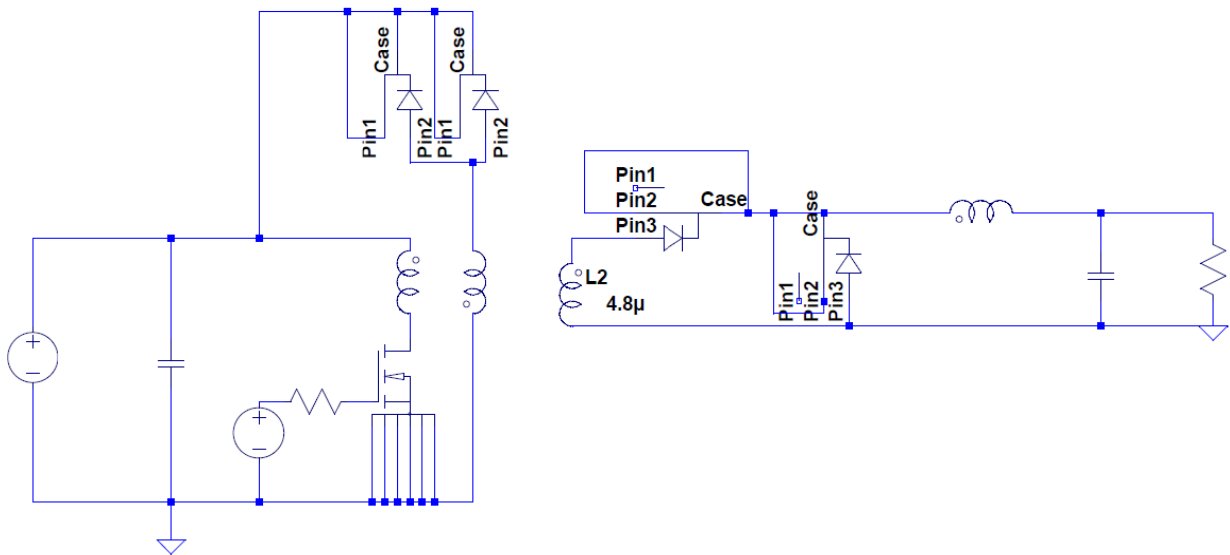Date: 06/03/16

200W FORWARD CONVERTER LTSPICE MODEL

REV 0

DISCLAIMER

Models provided by Wolfspeed are not warranted by Wolfspeed 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. Wolfspeed will not be responsible for any error or simulation issue arising due to the editing of the model library file.

Date: 06/03/16

This document is prepared as a quick reference guide to perform simulations based on Wolfspeed SiC power MOSFET PSPICE library using LTSPICE simulation software.

MODEL SPECIFICATIONS

SPECIFICATION OF CONVERTER:

Vin = 115Vdc

Vout = +12Vdc

Pout = 500W

Switching Frequency = 100 KHz

MODEL LIMITATIONS

- This converter model is purely simulating the performance of the switching device of Wolfspeed products and without considering others parasitic from capacitor and transformer losses.
- No feedback control circuit has been developed.
- Refer to C3M LTSPICE User Guide for C3M0065090J component model limitation [1].

ABOUT MULTIPLE OUTPUT FLYBACK CONVERTER LTSPICE MODEL

This LTspice model is created based on above converter specification. The intention of creating this Forward converter is to provide customers a sample reference circuit and a quicker way to evaluate the performance of Wolfspeed SiC MOSFET and diode. This model allow user to modify some of the components value to achieve a different design requirement.

PREREQUISITE:

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

MOSFET SPICE PACKAGE:

- SPICE Library Packaged Device Model (C3M0065090x_model_library_1p0.lib) – Model includes the D2Pak package parasitic [1].

- LTSPICE Device Symbol (nmos7.asy)

DIODE SPICE PACKAGE:

- SPICE Library Packaged Device Model (Cree Power C3D Packaged SPICE Model Library.lib) – Model includes TO-220 package parasitic [2].

- SPICE Library Packaged Device Model (Cree Power C5D CVFD Packaged SPICE Model Library.lib) – Model includes TO-220 package parasitic [2].

- LTSPICE Device Symbol (C3D08060A.asy, C5D50065D.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 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.

Date: 06/03/16

2. Copy the 500W Forward converter model file and paste it into the LTSPICE directory or any folder that user normally used. Typical path is given by (C:\Program Files (x86)\LTC\LTspiceIV\). This would make the model appear in the open window.

3. The model will be similar to the one shown in figure 1.

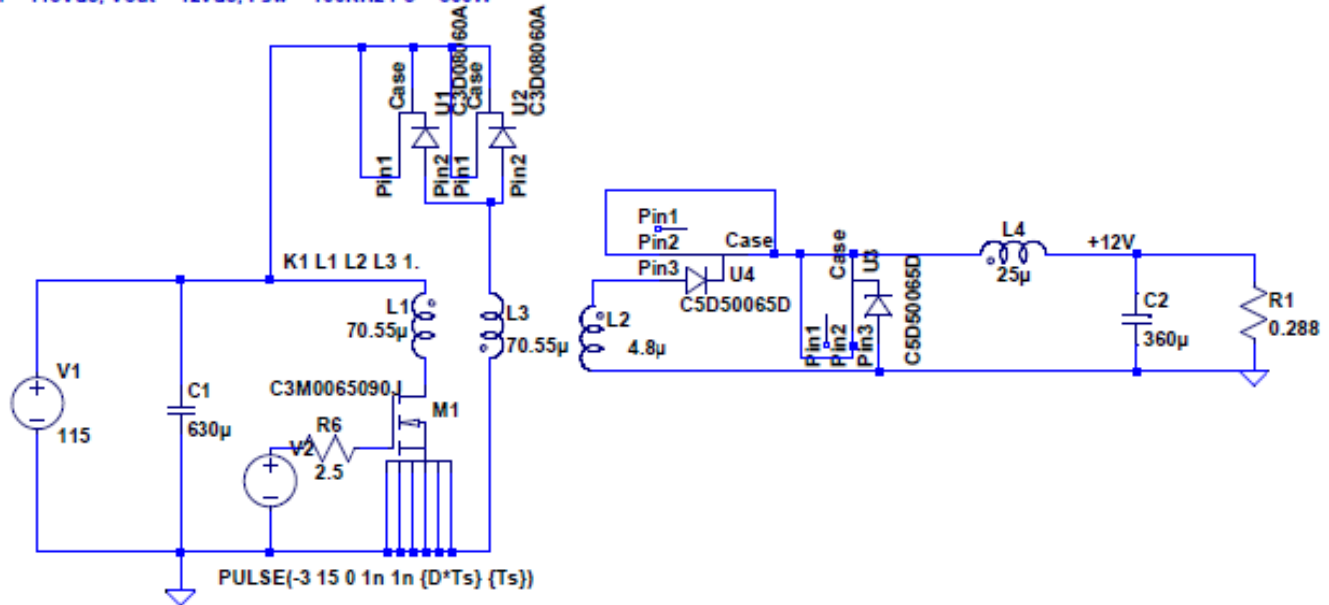

Figure 1: 500W Forward converter model circuit

4. This model allows user to change the value of some components to meet their design requirement.

SIMULATION GUIDELINES:
In case user has difference design requirement, below are the component value at converter LTSPICE model that allows user to change in order to produce the desired output:
- Vin (V1)
- Vout (Voltage across R1)
- D is duty cycle
- Ts is switching period
- Pulse signal (V2) for MOSFETs M1
- Output load (R1)

While performing simulations, it is not recommended to change simulation settings of the following LTSPICE directive which is already optimized base on the simulation speed and avoid any convergence error.

.options Gmin=1e-7 Abstol=1e-7 reltol=1e-2 chgtol=1e-9

Date: 06/03/16

RESULTS

Description	Design Requirement	LTspice results	SpeedFit Results ***	Experimental Results
V_{in} (V)	115	115	N.A.	N.A.
V_o (V)	12	12	N.A.	N.A.
f_{sw} (KHz)	100	100	N.A.	N.A.
P_o (W)	500	500	N.A.	N.A.
PLoss**(W)		70.94	N.A.	N.A.
Efficiency (%)*		87.57	N.A.	N.A.

Table 1 LTspice result against design requirement

* Efficiency is only based on semiconductor only.

**PLoss is the total semiconductor loss which is conduction and switching losses.

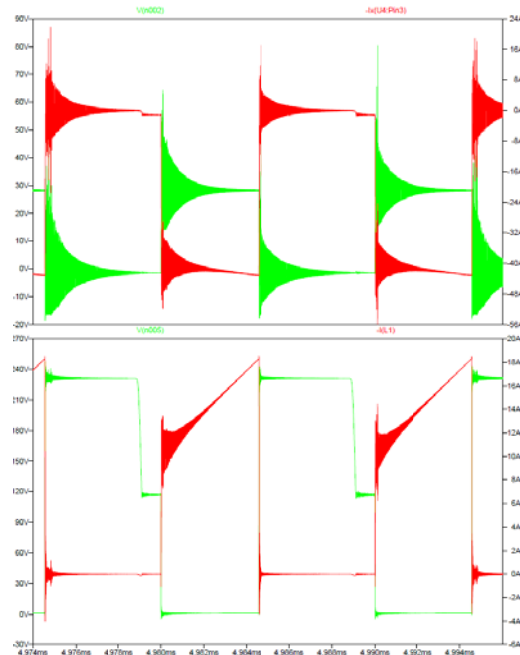***** SpeedFit is a Wolfspeed online simulation tool.[3]**

Figure 2: Converter LTspcie Output waveforms

* Note: (Top Green: Vd_U3;Top Red: Id_U4;Bottom Green: Vds_M1; Bottom Read: Ids_M1)

Date: 06/03/16

REFERENCE

[1] Wolfspeed MOSFET LTspice models. Citing Websites. From http://go.wolfspeed.com/MOSFETModel
[2] Wolfspeed Diode LTspice models. Citing Websites. From http://go.wolfspeed.com/diodeModel
[3] SpeedFit. Citing Websites. From http://www.wolfspeed.com/speedfit/