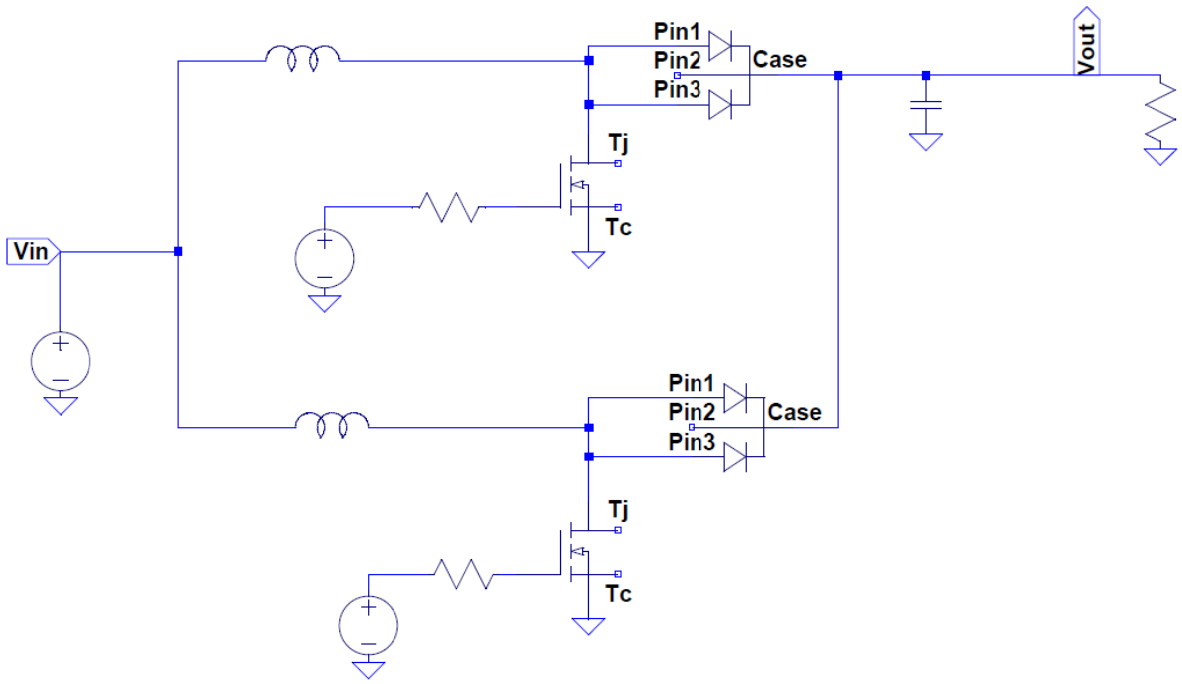
Date: 06/02/16

10KW INTERLEAVED BOOST CONVERTER LTSPICE MODEL

REV 1



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.

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

Date: 06/02/16

MODEL SPECIFICATIONS
SPECIFICATION OF CONVERTER:
Vin = 450Vdc
Vout = 640Vdc
Pout = 10KW
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 C2M LTSPICE User Guide for C2M0080120D component model limitation [2].

ABOUT ZVS LLC RESONANT FULL BRIDGE CONVERTER LTSPICE MODEL
This LTspice model was created based on the 10KW Interleaved Boost converter reference design [1]. The intention of creating this converter circuit is to provide customers a sample reference circuit and also a quicker way to evaluate the performance of the Wolfspeed SiC MOSFET and diode. This LTspice model offers a convenient starting point for users 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 (C2M0080120D - Packaged .lib) – Model includes the TO-247 package parasitic [2].
- LTSPICE Device Symbol (power_nmos_heat.asy)

DIODE SPICE PACKAGE:

- SPICE Library Packaged Device Model (Cree Power C3D Packaged SPICE Model Library.lib) – Model includes TO-220 package parasitic [3].
- LTSPICE Device Symbol (C4D10120D.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.

2. Copy the 10KW Interleaved Boost 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.

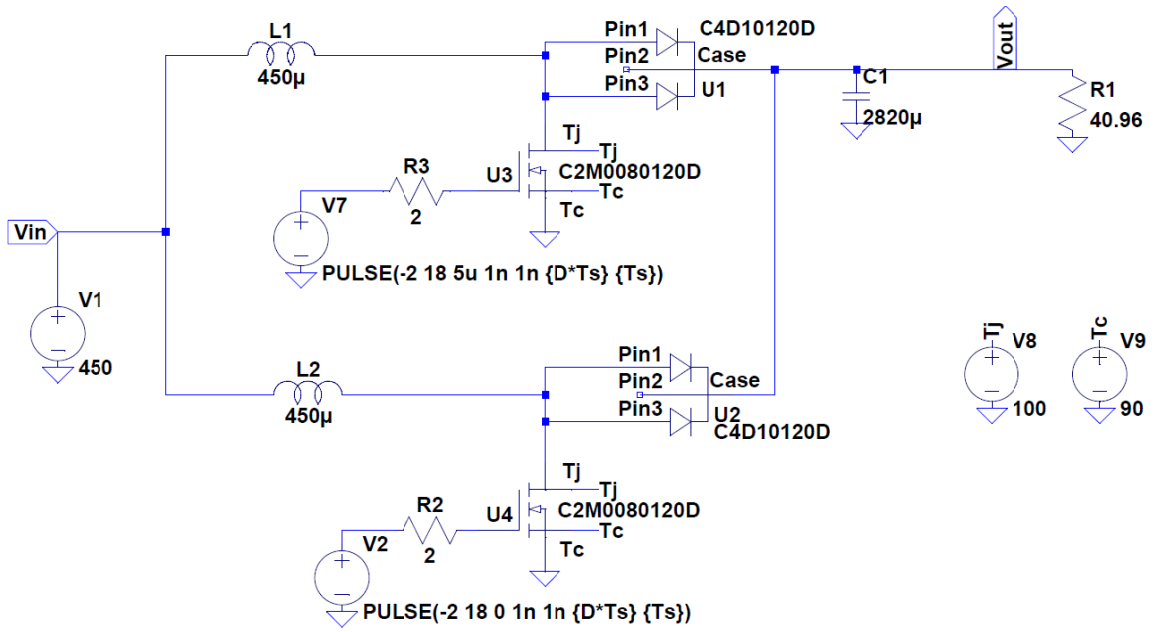


Figure 1: 10KW Interleaved Boost converter model circuit

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)
- Pulse signal (V2) for MOSFETs U3
- Pulse signal (V7) for MOSFETs U4
- D is duty cycle
- Ts is switching period
- Tj source (V8)
- Tc source (V9)
- Output load (R1)

The terminals Tj and Tc are representing the temperature of the junction and case of the MOSFET. The temperature connections are working as voltage pins. Therefore a potential difference of 1V refers to a temperature difference of 1°C. User can vary these values base on your own cooling design and observe the performance of the MOSFET or inverter.

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: 06/02/16

RESULTS

Description	Design Requirement	LTspice results	SpeedFit Results ***	Experimental Results
V_{in} (V)	450	450	N.A.	450
V_o (V)	640	638.42	N.A.	640
f_{sw} (KHz)	100	100	N.A.	100
P_o (KW)	10	9.95	N.A.	10
PLoss**(W)		48.61	N.A.	57.35
Efficiency (%)*	>99	99.28	N.A.	99.2

Table 1 LTspice result against design requirement and experimental result

*** Efficiency has included the magnetic loss of 23.3W from the paper for easier comparison.**

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

***** SpeedFit is a Wolfspeed online simulation tool.[4]**

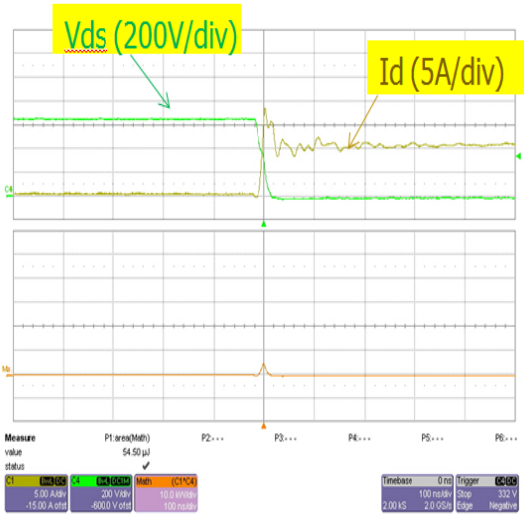


Figure 2: Converter experimental waveform

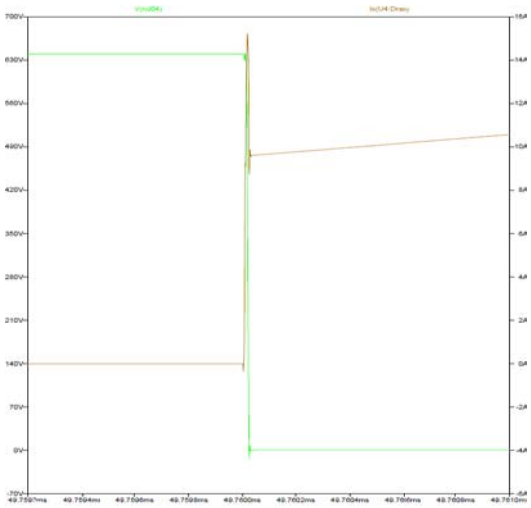


Figure 3: Converter LTspice waveform

Note: (Yellow: I_diode; Green: Vds)

REFERENCE

[1] Jimmy Liu, "Increase Efficiency and Lower System Cost with 100KHz, 10kW Silicon Carbide (SiC) Interleaved Boost Circuit Design"
[2] Wolfspeed MOSFET LTspice models. Citing Websites. From http://go.wolfspeed.com/MOSFETModel
[3] Wolfspeed Diode LTspice models. Citing Websites. From http://go.wolfspeed.com/diodeModel
[4] SpeedFit. Citing Websites. From http://www.wolfspeed.com/speedfit/