Wolfspeed SiC MOSFET LTspice Model
Quick Start Guide

Power Applications

Rev2.0

Wolfspeed SiC MOSFET LTspice Model Quick Start Guide Rev 2.0, 02-2018

Table of Contents

DISCLAIMER

Models provided by WOLFSPEED are not warranted by WOLFSPEED, CREE 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. CREE 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 it's 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 into the market. WOLFSPEED MOSFET LTspice models contain CXM0XXXXXXX and CPMX-XXXX-XXXXX electro-thermal LTspice models for the packaged device and bare die. LTspice models included a provision of self-heating to observe the change in junction temperature of the device. These LTspice models provide a reasonable approximation for the MOSFET in the third quadrant. However, the body diode threshold voltage was modeled at V_{GS} = -4V (Gen 3) or -5V (Gen2) and assumes this is fixed for all values of V_{GS}. However, the effect of a slight change in the turn-ON voltage of the body diode over the range of $-4V \leq V_{GS} \leq 0V$ is not modeled. 3rd quadrant of the MOSFET is optimized and verified for T_c=25°C & 150°C at V_{GS} = 15V (Gen 3) or 20V (Gen2). The Model is most accurate at the I_{DS} (DC) @ T_c=25°C & 150°C operating conditions as shown in the device data sheet.

2. LTspice Software

2.1 Prerequisite:

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

2.2 Package Contents:

- SPICE Library Packaged Device Model (CXM0XXXXXXXD.lib) – Device model includes TO-247 3Leads package.
- SPICE Library Packaged Device Model (CXM0XXXXXXXK.lib or CXM0XXXXXXXP.lib) – Device model includes TO-247 4Leads package.
- SPICE Library Packaged Device Model (CXM0XXXXXXXJ.lib) – Device model includes TO-263 7Leads package.
- SPICE Library Bare Die Model (CPMX-XXXX-XXXXX.lib) – Die model does not include any package parasitic.
- LTSPICE Device Symbol (nmos TO247_3L.asy, nmos TO247_4L.asy & nmos TO263_7L.asy)
- LTSPICE Die Symbol (nmos_die.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.

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 MOSFET 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. LTspice provides the option for changing the visibility of the labels associated with the terminals.

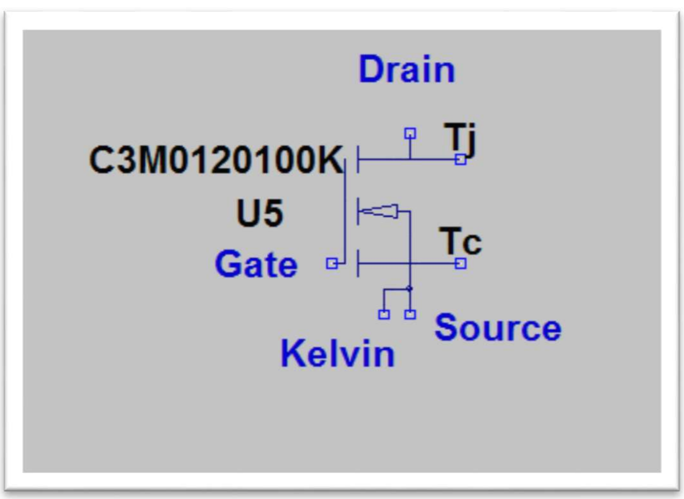

Figure 1. Device symbol in a schematic (reference purpose only).

6. Copy the Wolfspeed MOSFET 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 25°C & 150°C temperature and gate to source voltage (V_{GS}) of 15V (Gen 3) or 20V (Gen2). Modest accuracy on the rest of Vgs for 25°C & 150°C.
- Valid for temperature range -55°C to 150°C
- Optimized and verified for 25°C & 150°C temperature and gate to source voltage (V_{GS}) of 15V (Gen 3) or 20V (Gen2) for 3rd Quadrant only.
- Body diode operation optimized for V_{GS} = -4V (Gen 3) or -5 (Gen 2)
- Model includes self-heating and transient thermal capability.
- Parasitic inductance associated with electrodes will be included in the model.

3.2 Model Limitation

- Parasitic BJT and its associated effects not modeled.
- Avalanche multiplication process not modeled.
- Variation of body diode turn-ON voltage with gate to source voltage is not modeled.

4. Simulation Guidelines:

The SiC LTspice model is provided with the following terminals:

- Drain
- Gate
- Kelvin (Except on TO-247- 3 lead package & die model)
- Source
- Junction Temperature terminal - Tj
- Case Temperature – Tc (Except on die models)

The same symbol can be used for different MOSFET models if the pin count is the same. For example nmos TO247_4L.asy can be used for MOSFET model C3M0065100K, C3M0120100K or any

MOSFETs with 4 leads. It is also applied to nmos7.asy for MOSFET model C3M0065100J, C3M0120090J or any MOSFETs with 7 leads. To achieve that, move the mouse pointer to the symbol and right click. Then component attribute editor will appear, enter the MOSFET model number at the attribute “Value” like figure 2.

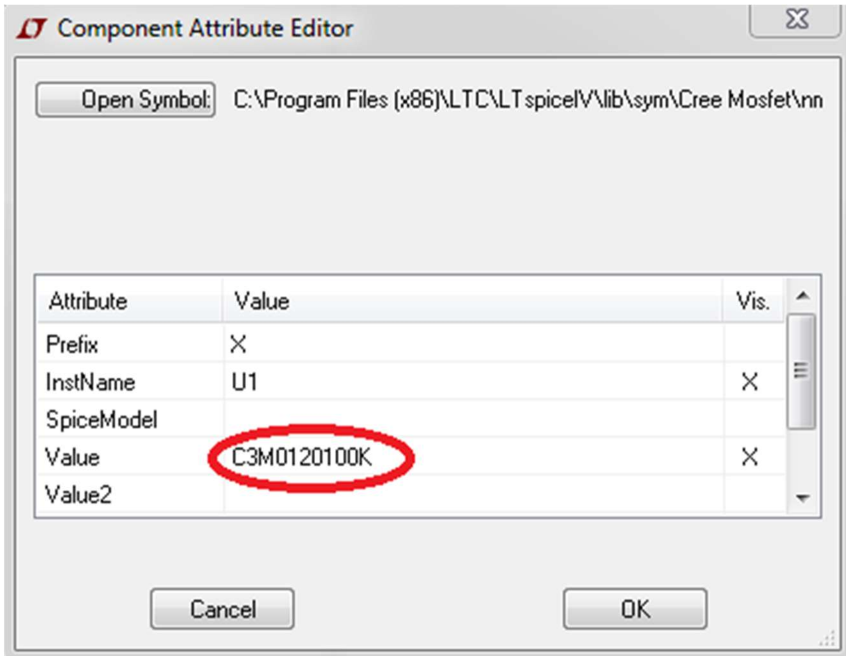

Figure 2: Component Attribute Editor

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\C3M0120100K.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 3 shows the connection of Tc terminal to an ambient temperature of 25°C.

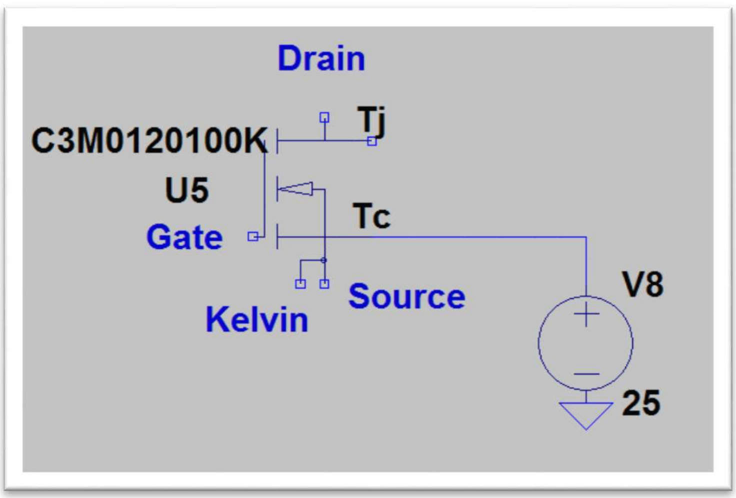

Figure 3: Fixing Case Temperature to 25°C

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.

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 4.

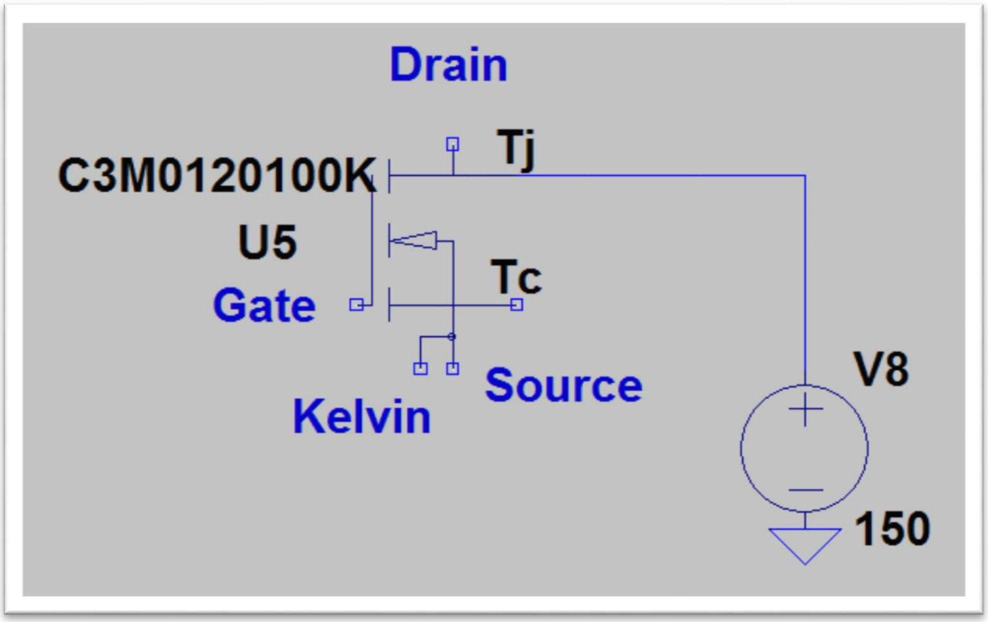

Figure 4: Junction maintained at a constant temperature of 150°C for DC simulation.

5. Migrating Wolfspeed LTspice model to others SPICE softwares

Wolfspeed MOSFET SPICE models are both LTspice and OrCad Pspice compatible. To use Wolfspeed MOSFET SPICE model on other SPICE softwares, user may need to do few more steps to get it work. 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 uses their own symbol format thus user should create their own symbol.

Note: It is the responsibility of user to verify the model against datasheet after changing the format of the model.

6. Simulation Examples

6.1 Simulation Example1:

The schematic in Figure 5 shows the LTspice model of the Boost converter. The purpose of this example is to show the connection of Kelvin source, Junction temperature terminal and including the library path.

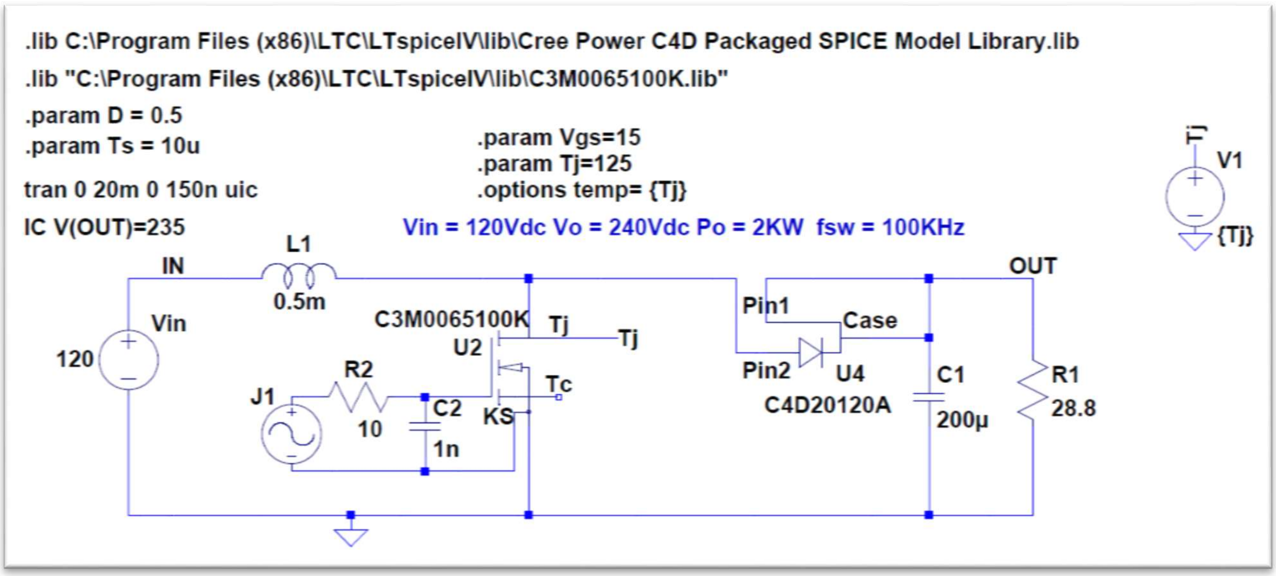

Figure 5. LTspice model of the Boost converter

Waveforms of Drain Voltage (Green) and Drain Current (Red) are shown in figure 6, 7 & 8.

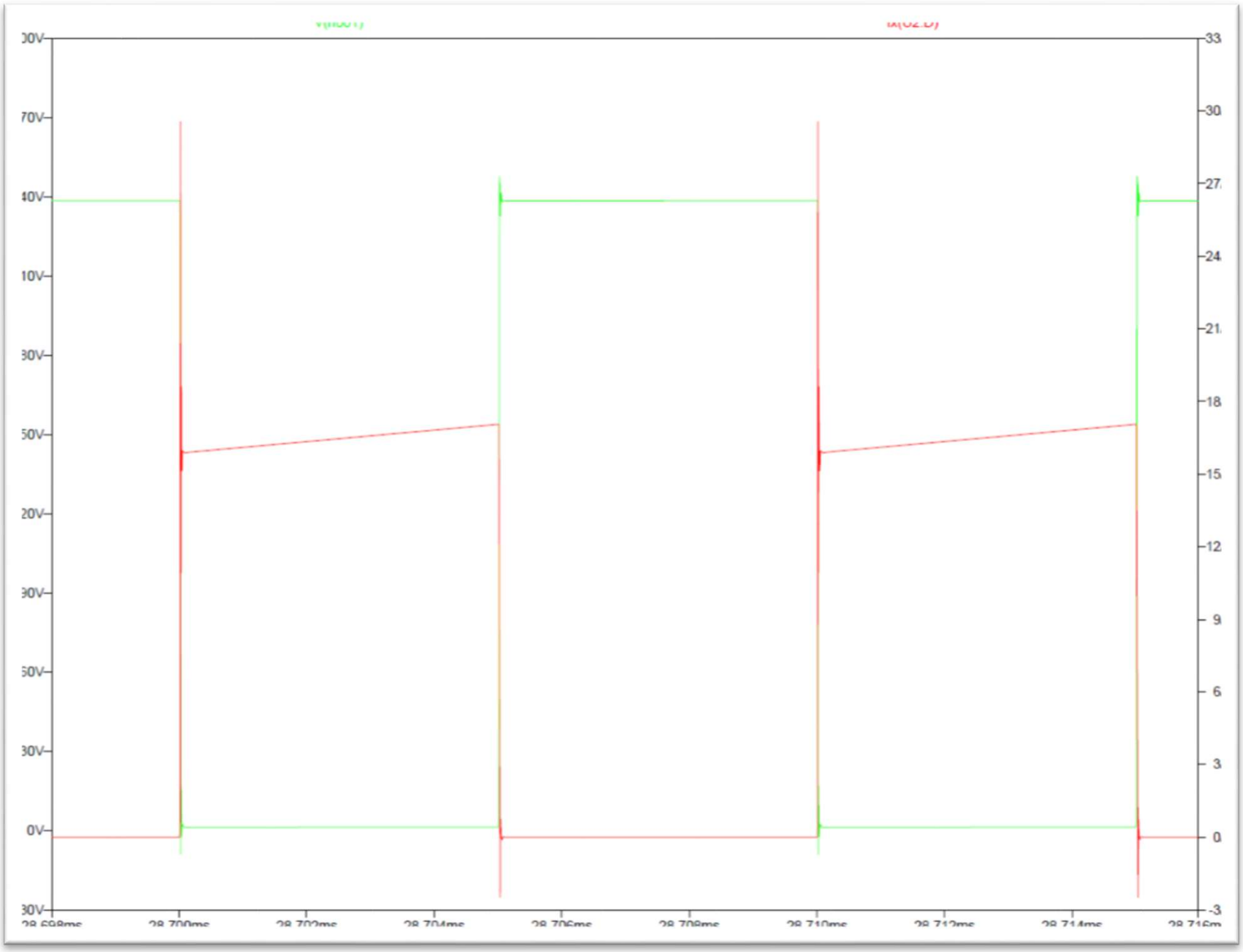

Figure 6. Waveform Screenshot obtained from LTspice Simulation of Boost converter

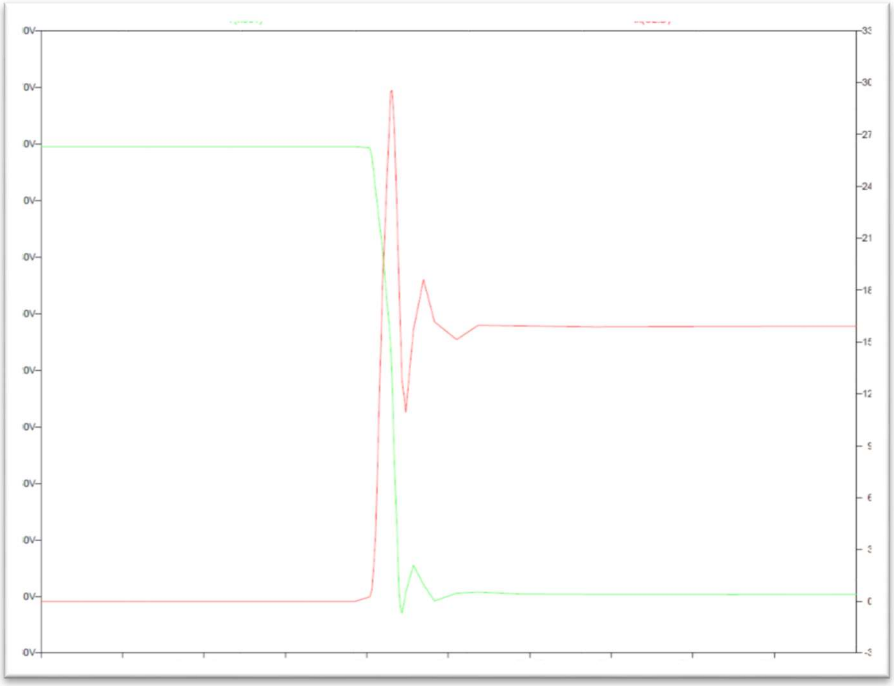

Figure 7. SiC MOSFET Turn-ON Event

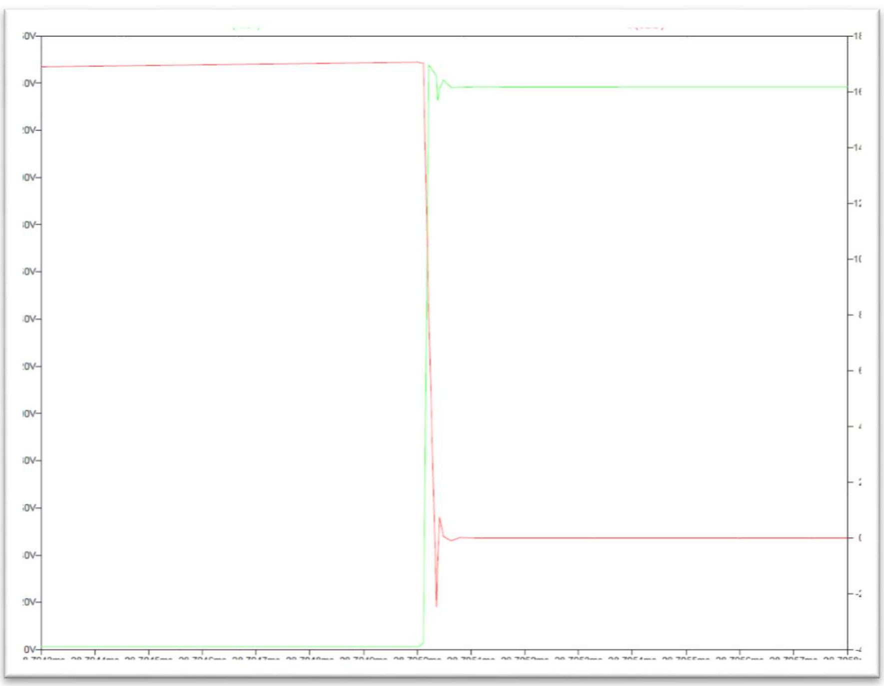

Figure 8. SiC MOSFET Turn-OFF Event

6.2 Simulation Example 2:

Example 2 shown in figure 9 is a 3 phase 2 level inverter with output filter. The purpose of this example is to show the connection of Kelvin source of TO263-7, and gate drive.

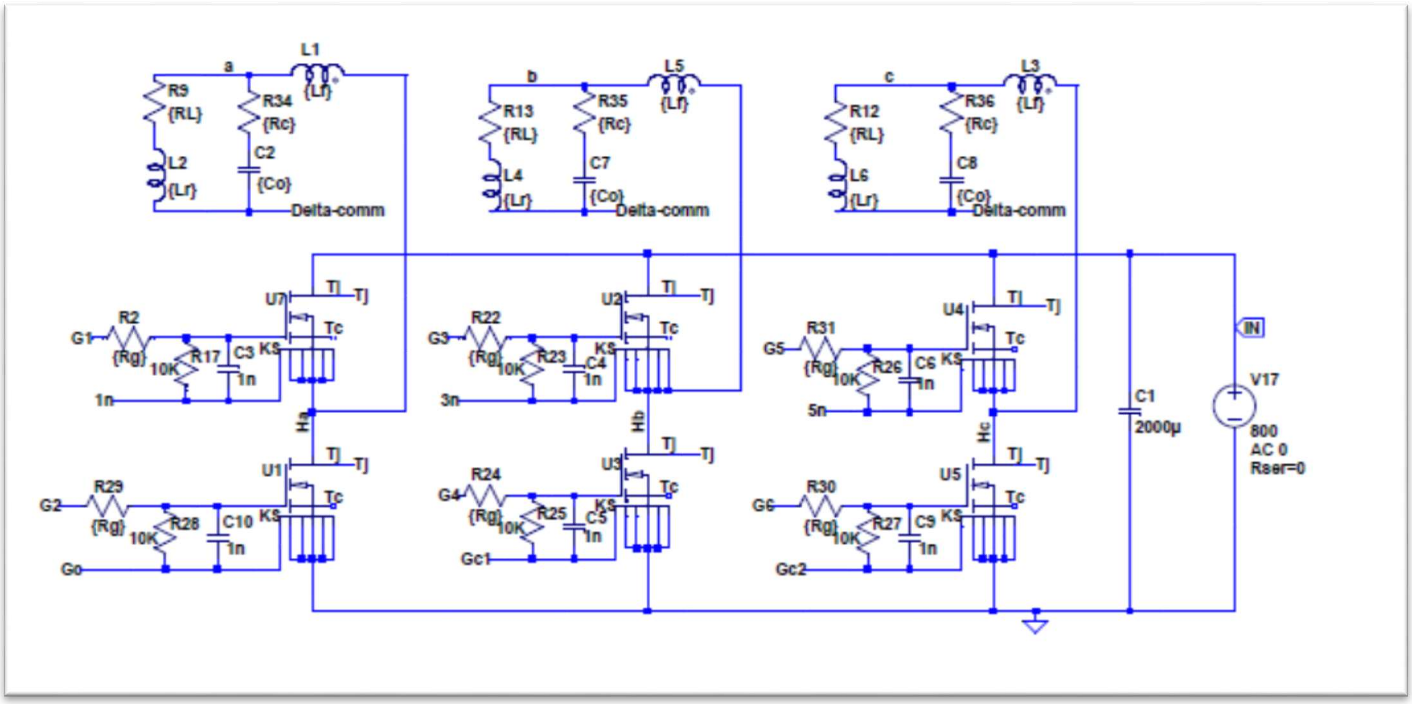

Figure 9. 3phase 2Level Inverter with output filter

Output inverter phase a voltage (Blue) and current (Green) waveforms are shown in figure 10.

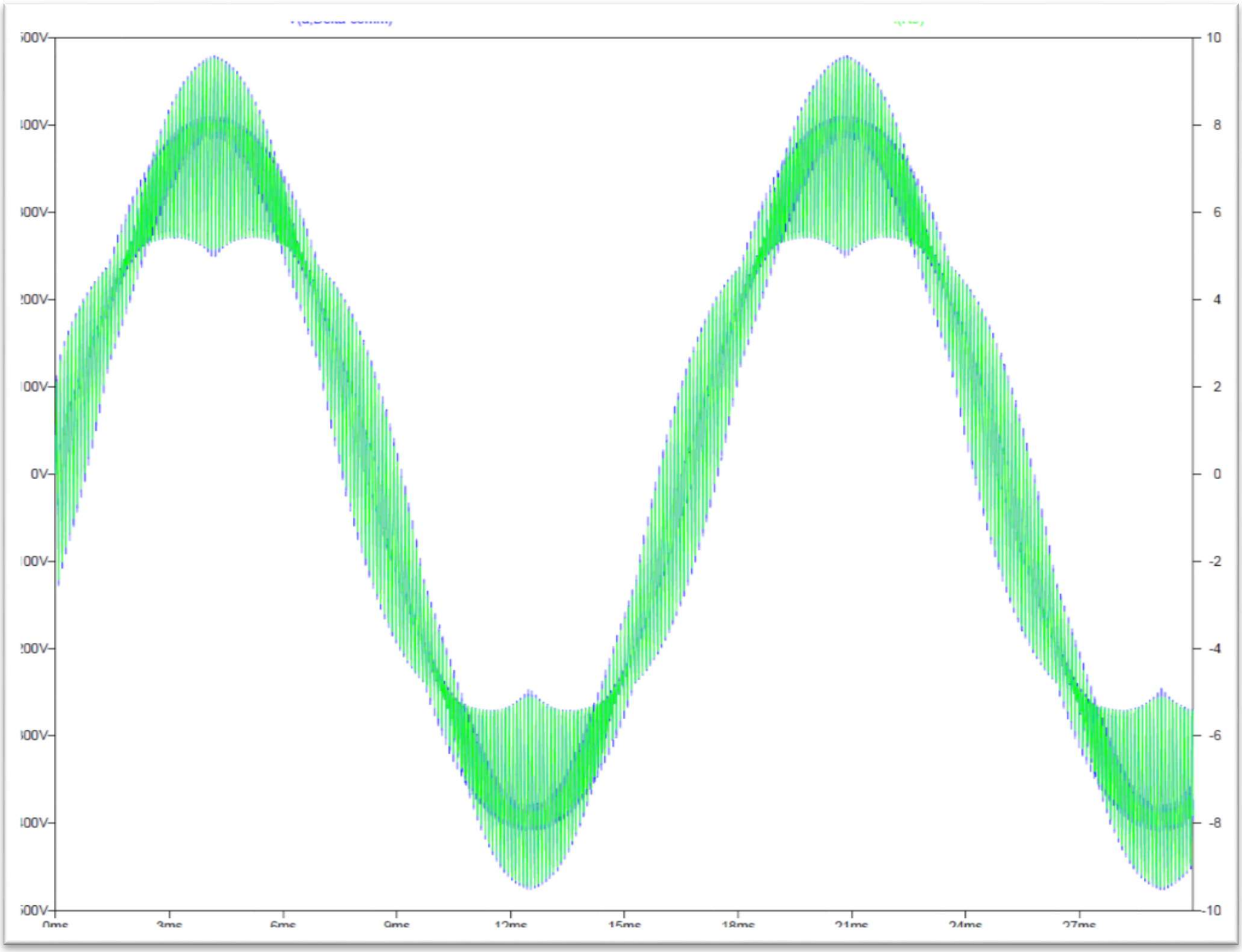

Figure 10. Waveform captured from LTspice Simulation of 3phase inverter

7. Revision History

Date	Revision	Changes
12/21/2017	V1.0	Initial release
02/21/2018	V2.0	SPICE model is both LTspice and Pspice compactible