



Wolfspeed SiC MOSFET SPICE Model Quick Start Guide

Power Applications

Rev1.0

Table of Contents

1. Introduction.....	3
2. OrCad Pspice Software.....	3
2.1 Prerequisite.....	3
2.2 Package Contents.....	3
2.3 Software Requirement	4
2.4 Model Installation Guidelines.....	4
3. Model Specifications.....	5
3.1 Model features.....	5
3.2 Model Limitation.....	5
4. Simulation Guidelines.....	5
5. Migrating Wolfspice SPICE model to others SPICE software.....	8
6. Simulation Examples.....	9
6.1 Simulation Example1.....	9
7. Revision History	12

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 SPICE model is to allow the users to create a converter circuit and understand its design and performance parameters. With the simulation of SPICE 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 SPICE models contain CXM0XXXXXXX and CPMX-XXXX-XXXX electro-thermal SPICE models for the packaged device and bare die. SPICE models included a provision of self-heating to observe the change in junction temperature of the device. These SPICE 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 $T_c=25^\circ C$ & $150^\circ C$ at $V_{GS} = 15V$ (Gen 3) or $20V$ (Gen2). The Model is most accurate at the I_{DS} (DC) @ $T_c=25^\circ C$ & $150^\circ C$ operating conditions as shown in the device data sheet.

2. OrCad Pspice Software

2.1 Prerequisite:

OrCad Pspice simulation software

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-XXXX.lib) – Die model does not include any package parasitic.
- PSPICE Device Symbol (Wolfspeed-TO247-3L.OLB, Wolfspeed-TO247-4L.OLB & Wolfspeed-TO263-7L.OLB)
- PSPICE Die Symbol (Wolfspeed-Die.OLB)

2.3 Software Requirement:

This model has been developed and optimized for Pspice. It is the responsibility of the user to be well-versed with the basic operation of Pspice 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 or verify the result before use.

2.4 Model Installation Guidelines:

1. Download the SPICE 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 (.OLB) and paste it into the Pspice symbol directory. Typical installation path is given by (C:\Cadence\SPB_XX\tools\capture\library\pspice\modeled\). 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. Pspice provides the option for changing the visibility of the labels associated with the terminals.

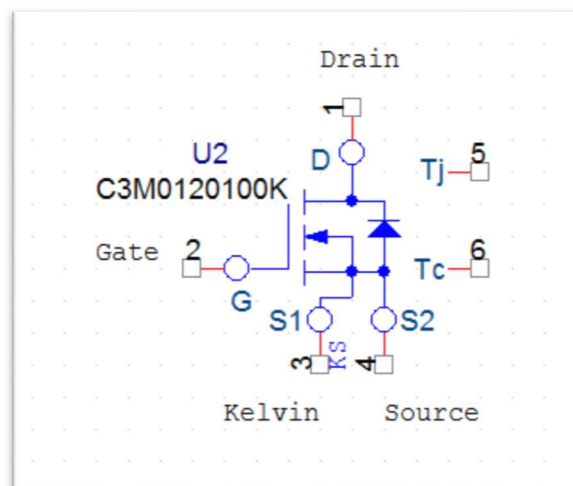


Figure 1. Device symbol (reference purpose only).

6. Copy the Wolfspeed MOSFET library file (.lib) and paste it into the Pspice library directory. Typical installation path is given by (C:\Cadence\SPB_XX\tools\pspice\library\Wolfspeed diode).

3. Model Specifications

3.1 Model features

- Optimized for 25°C & 150°C temperatures and gate to source voltage (V_{GS}) of 15V (Gen 3) or 20V (Gen2). Modest accuracy on the rest of V_{GS} for 25°C & 150°C temperatures.
- Valid for temperature range -55°C to 150°C
- Optimized and verified for 25°C & 150°C temperatures 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)
- The 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 (V_{GS}) is not modeled.

4. Simulation Guidelines:

The SiC Pspice model is provided with the following terminals:

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

The same symbol consists of different MOSFET models if the pin count is the same. For example, Wolfspeed-TO247_4L.OLB consists of MOSFET models like C3M0065100K, C3M0120100K or any

MOSFETs with 4 leads. It is also applied to Wolfspeed-TO263-7.OLB symbol for MOSFET model C3M0065100J, C3M0120090J or any MOSFETs with 7 leads. Once the symbol is selected from libraries, a list of available MOSFETs with their part numbers will be shown on the part list to allow the user to pick a MOSFET to perform the simulation.

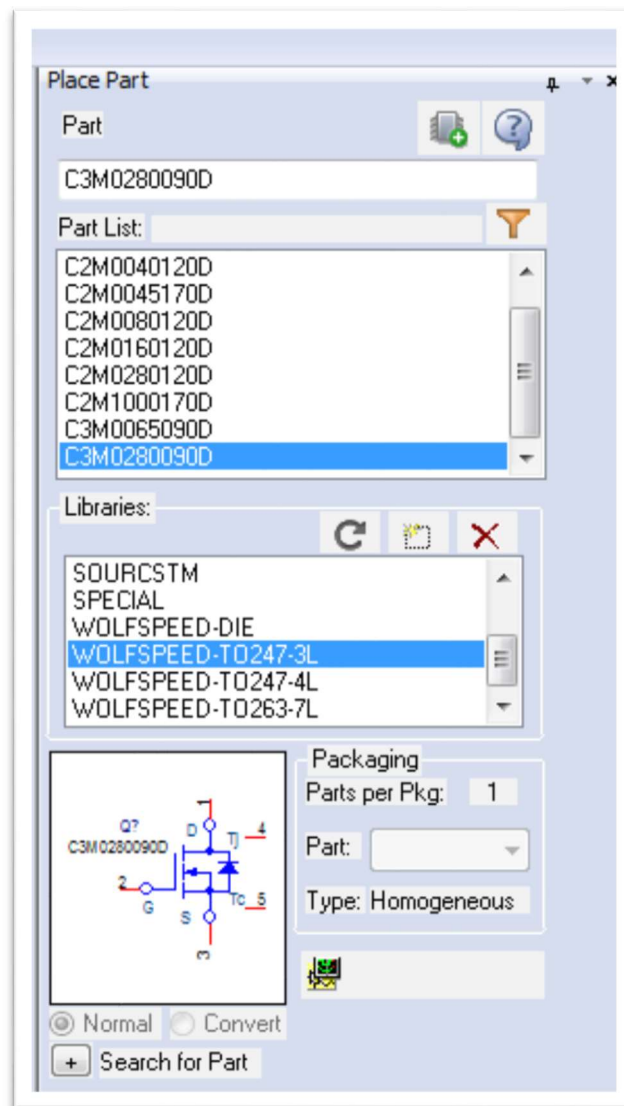


Figure 2: Wolfspeed-TO247-3L with a list of available 3 leads MOSFETs

The user is required to include the library path at each circuit design. This is to provide Pspice the path where the library is located and it can be done at “simulation profiles”.



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.

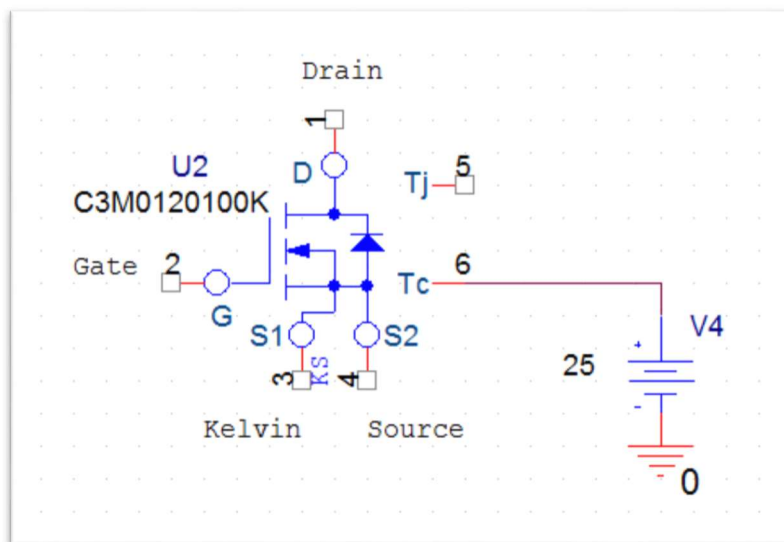


Figure 3: Fixing Case Temperature (Tc) to 25°C



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.

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.

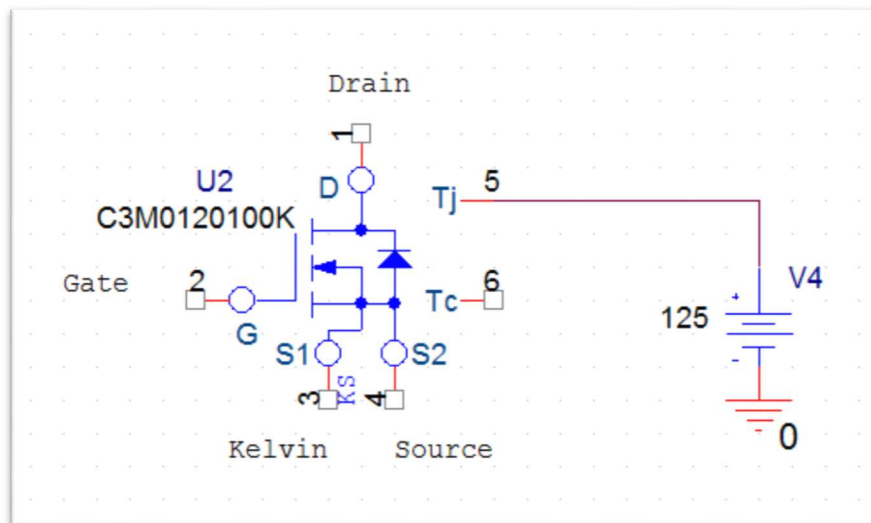


Figure 4: Junction maintained at a constant temperature of 125°C for DC simulation.

5. Migrating Wolfspice SPICE model to others SPICE software

Wolfspice MOSFET SPICE models are both LTspice and OrCad Pspice compatible. To use Wolfspice MOSFET SPICE model on other SPICE software, the user may need to do few more steps to get it work. Some SPICE software uses different extension like SIMetrix uses the library with extension .lb whereas Pspice and LTspice 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 the 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 Pspice 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.

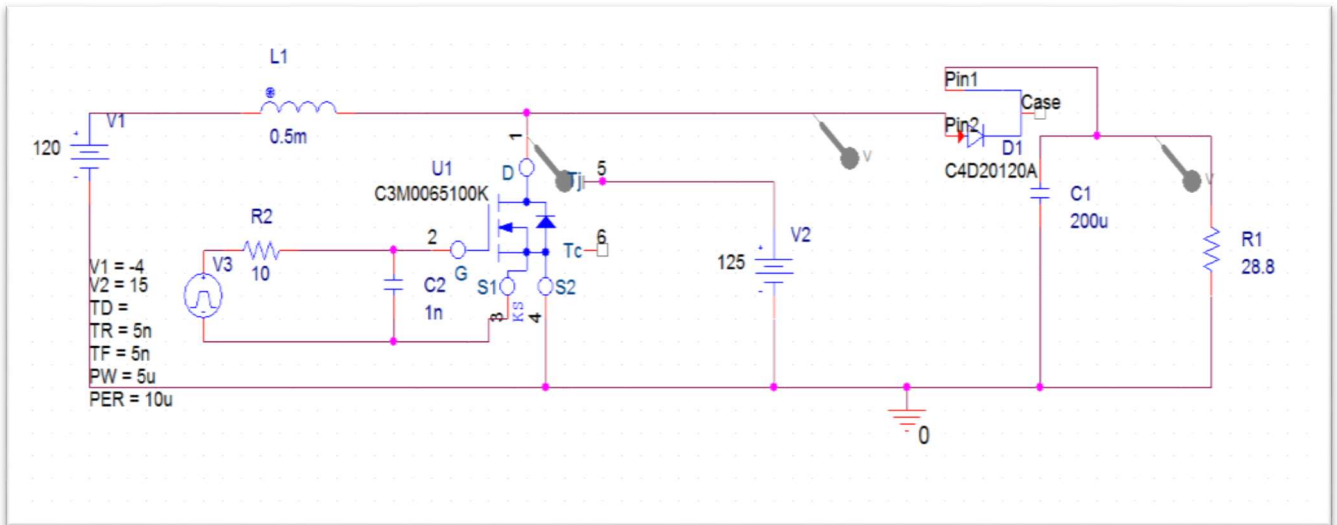


Figure 5. PSPICE model of the Boost converter



If encountered convergence error, the user can enable the Advanced Convergence and Auto Converge simulation setting. If the error persists, adding capacitance between 1E-14 to 1E-12 at cshunt will help to solve the issue. It is the responsibility of the user to ensure that the addition of small capacitance has not affected the simulation result.

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

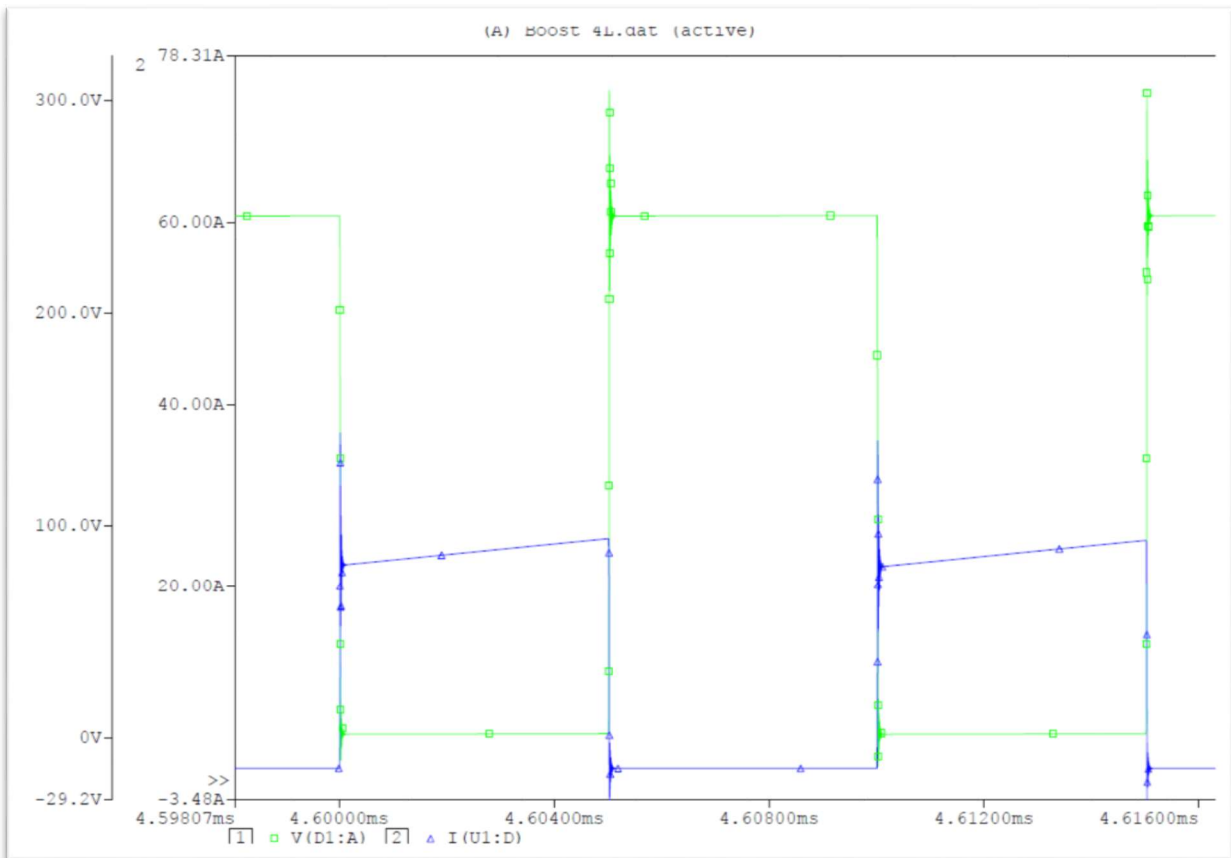


Figure 6. Waveform Screenshot obtained from Pspice Simulation of Boost converter

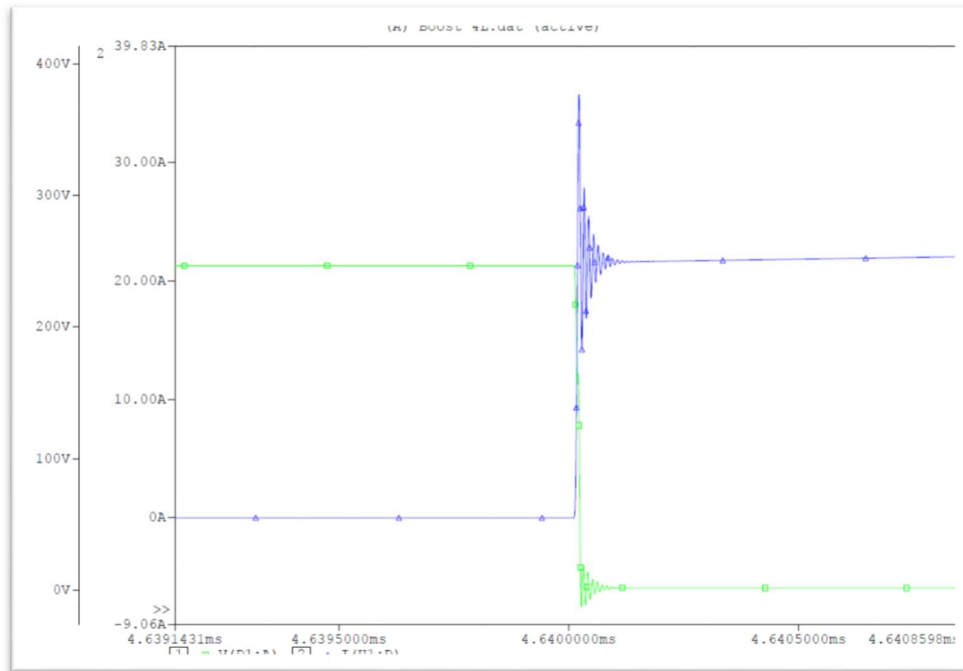


Figure 7. SiC MOSFET Turn-ON Event

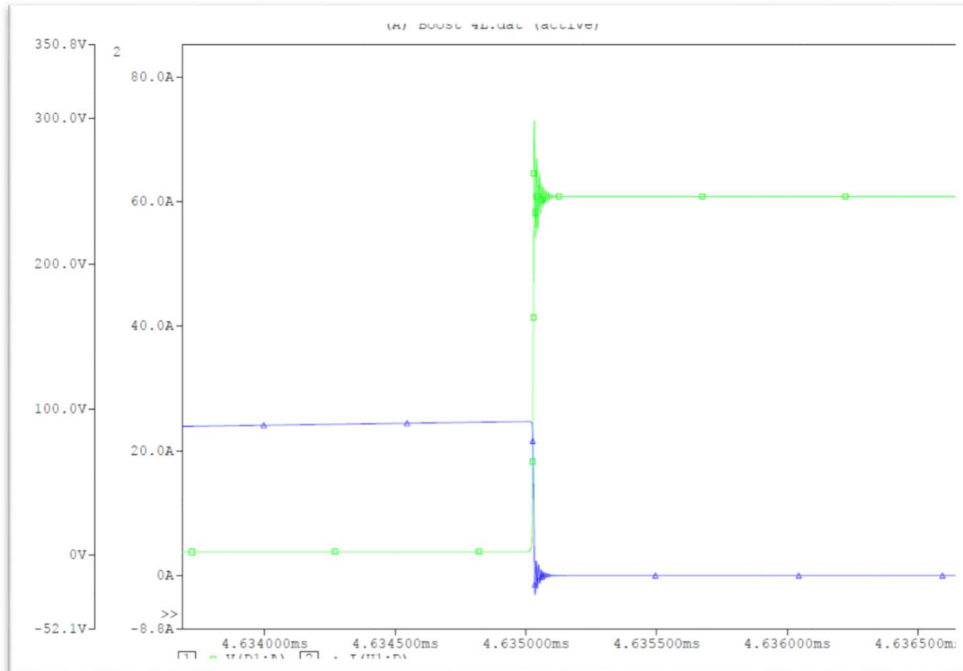


Figure 8. SiC MOSFET Turn-OFF Event

7. Revision History

Date	Revision	Changes
02/21/2018	V1.0	Initial release