Visit [www.gansystems.com](http://www.gansystems.com) for the latest version of this document
1. Introduction

Models provided by GaN Systems Inc. are not warranted by GaN Systems Inc. as fully representing all of the specifications and operating characteristics of the semiconductor product to which the model relates. The models describe the characteristics of typical devices. 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 breadboarding for final verification. GaN Systems Inc. therefore does not assume any liability arising from their use. GaN Systems Inc. reserves the right to change models without prior notice.

2. Definitions and Functions of Model Levels

GaN Systems Inc. provides three different levels of models for GaN E-HEMT transistors. The definition and functions of the model levels are shown in the Table 1 and Table 2, respectively.

<table>
<thead>
<tr>
<th>Suffix</th>
<th>Level</th>
<th>Terminals</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>_L1</td>
<td>1</td>
<td>G, D, S, SS (if applicable)</td>
<td>General electrical simulations on application/converter level circuits. Focus on simulation speed.</td>
</tr>
<tr>
<td>_L2</td>
<td>2</td>
<td>G, D, S, SS (if applicable), Tc, Tj</td>
<td>In addition to L1, L2 also includes a thermal model. Self-heating temperature rise is also taken into account.</td>
</tr>
<tr>
<td>_L3</td>
<td>3</td>
<td>G, D, S, SS (if applicable), Tc, Tj</td>
<td>In addition to L2, L3 also includes the package stray inductance.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Functions</th>
<th>Level 1</th>
<th>Level 2</th>
<th>Level 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>IV performance as a function of temperature</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Voltage-dependent capacitance</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Thermal model</td>
<td>✗</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Package stray inductance</td>
<td>✗</td>
<td>✗</td>
<td>✓</td>
</tr>
</tbody>
</table>
2.1. Level 1 model

The Level 1 model has either three or four terminals. For those devices with a source sense terminal, a fourth pin SS will be provided as well. Their symbols are shown in Fig. 1. Note that the dual gates or dual source senses are not modelled.

![Level 1 model symbol](image)

Fig. 1. Level 1 model symbol, (a) without SS terminal, (b) with SS terminal.

For the Level 1 model, as the thermal network is not included, the junction temperature needs to be defined. User can put an additional SPICE Directive command to set the junction temperature as shown in below,

```
.option temp=85; Set Tj
```

2.2. Level 2 model

The Level 2 model has two additional pins. They are the Tc terminal (case temperature) and Tj terminal (junction temperature). The Level 2 symbol with a source sense terminal is shown in Fig. 2.

![Level 2 model symbol with SS terminal](image)

Fig. 2. Level 2 model symbol with SS terminal

The thermal model inside the device model is Cauer model [1]. The junction-to-case thermal resistance is included in the Spice model for each device. Therefore, user only needs to define either a case temperature or an ambient temperature along with adding a case-to-ambient thermal
resistance, like heatsink or thermal interface material (TIM). The thermal model structure and the possible connections are shown in Fig. 3. Unlike to Level 1, the TEMP does not need to be defined in the SPICE Directive command for Level 2.

**Note:** in the thermal model, voltage source indicates temperature in °C and resistor indicates thermal resistance in °C/W.

Fig. 3. SPICE Cauer model with (a) case temp as input, (b) case-to-ambient thermal resistance and ambient temp as inputs

A simple boost converter is used as an example shown in Fig. 4. An ambient temperature and heatsink thermal resistance are defined as 25°C and 10 °C/W, respectively. The simulation results are shown in Fig. 5 and Fig. 6. It is clear that the transient junction and case temperatures are risen from the 25°C ambient temperature and then reach to their steady states.
2.3. Level 3 model

In addition to Level 2 model, the Level 3 model also includes the package stray inductance as shown in Fig. 7a. The symbol of Level 3 is about the same to the Level 2 model, which is shown in Fig. 7b.
3. Model Files Setup

Each of the LTspice model files includes one .lib file and one .asy file. Before implementing the LTspice model-based simulation circuit, the LTspice model files need to be installed into the user's LTspice simulation tool library.

For LTspice model, the path to place the .lib file is shown as below,

This PC → Documents → LTspice → lib → sub

The path to place the .asy file is shown as below,

This PC → Documents → LTspice → lib → sym

Once place the files under the library, please restart the LTspice program. Then the device should be able to be found in the “component” (△) section.

4. Reference
