MOSFET

Renesas Power MOS SPICE model

About this document

This document will describe the Power MOS SPICE model provided by Renesas.

Target Device

Power MOSFETs

Contents

1. Introduction

In recent years, heat and noise caused by power devices have become a significant issue in the development of applications that incorporate power devices, and simulation is becoming more important to shorten development time. SPICE, a circuit simulator, can perform various calculations such as power loss calculation of power devices, prediction of gate drive performance, circuit avalanche energy, switching time, and transient junction temperature rise.

Since there is no standard SPICE model for power devices, power device manufacturers develop their own models and provide them to users. This paper describes the features of our SPICE model for Power MOSFETs that Renesas offers.

2. Power MOSFETs SPICE model

2.1 Supported Simulators

There are many types of commercial circuit simulators. We provide SPICE models for the following simulators (Table 2-1), which are mainstream in the field of power electronics.

Table 2-1 Supported Simulators

Name	Vendor
PSPICE	Cadence
LTspice	Analog Devices

2.2 Grades of SPICE model

As mentioned in the Introduction, SPICE models for power devices must be developed independently by the manufacturer. To obtain high simulation accuracy, SPICE models need to be complex, but this in turn reduces simulation speed and worsens calculation convergence. Therefore, we offer three models of different grades: the G1 model, which emphasizes calculation speed, the G2 model, which emphasizes accuracy, and the G3 model, which can analyze the effects of self-heating effects (Table 2-2).

Table 2-2 Grades of SPICE model

Grade	Encryption	Features
G1	Non-encryption	emphasizes calculation speed and convergence
G2	Encryption	emphasizes calculation accuracy
G3	Encryption	Self-heating effects can be calculated based on G2

2.3 Encryption

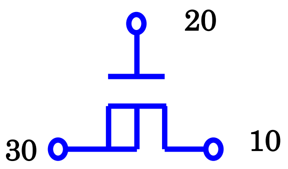
SPICE models contain confidential information such as modeling techniques to accurately represent device characteristics and device structures. Therefore, G2 and G3 models, which are high-precision SPICE models, are encrypted and published. Encrypted models can only be used on the target simulator and cannot be edited at all, which is not convenient for users. Therefore, for user convenience, the G1 model, which is a simple SPICE model, is published without encryption.

2.4 G1 model

The G1 model is a model of the basic operation of a MOSFET with a minimum number of elements. This model is best suited when the focus of the simulation is on speed.

(1) Terminal configuration

The G1 model consists of three terminals



Port	Description
10	Drain
20	Gate
30	Source

Figure 2-1 Terminal configuration (G1 model)

(2) Instance parameters

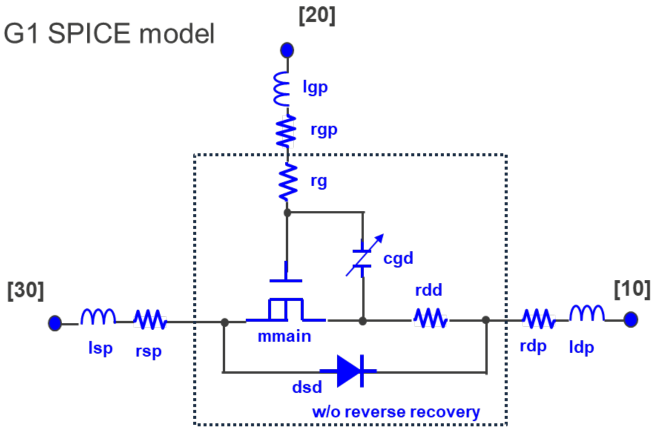
The following instance parameters are available for users to change:

dm: multiplier (dm=1 for target product). Used when you want to resize the target product.

Table 2-3 Instance parameters (G1 model)

Params	Default	Descriptions
dm	1	Multiplier

(3) Model



Parts	model
mmain	MOS3 (Level=3)
rg	Constant R
rdd	Constant R
cgd	Voltage variable C
dsd	Diode
rdp,ldp	PKG parasitics
rgp,lgp	PKG parasitics
rsp,lsp	PKG parasitics

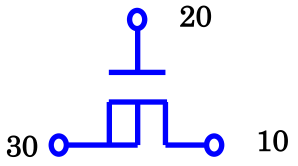
Figure 2-2 Model equivalent circuit (G1 model)

2.5 G2 model

To achieve high simulation accuracy, the G2 model uses the BSIM3 model in the MOS section, incorporating bias dependence in Cgd and Rdd and reverse recovery effects in body-diode. This model is ideal when simulation accuracy is important.

(1) Terminal configuration

The G2 model consists of three terminals



Port	Description
10	Drain
20	Gate
30	Source

Figure 2-3 Terminal configuration (G2 model)

(2) Instance parameters

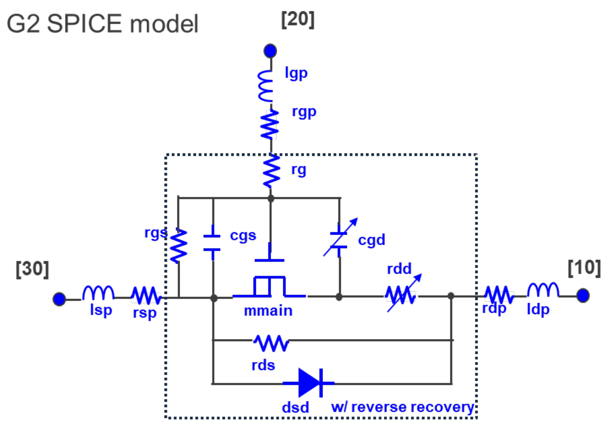
The following instance parameters are available for users to change:

ggmin: Minimum conductance. Effective only for the target element. Used when convergence is poor.

Table 2-4 Instance parameters (G2 model)

Params	Default	Description
ggmin	1e-15	Minimum conductance

(3) Model



Parts	model
mmain	BSIM3 (PSpice Level=7, LTspice Level=8)
rg	Constant R
rdd	Constant R
cgd	Voltage variable C
dsd	Diode macro model w/ reverse recovery
rdp,ldp	PKG parasitics
rgp,lgp	PKG parasitics
rsp,lsp	PKG parasitics

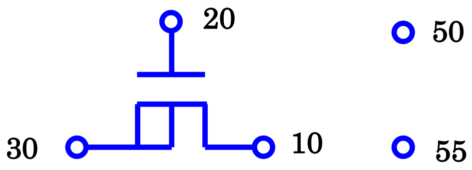
Figure 2-4 Model equivalent circuit (G2 model)

2.6 G3 model

The G3 model incorporates self-heating effects into the G2 model and can analyze the effects of dynamic temperature changes during simulation; by setting the case temperature as a voltage source at the Tc pin or connecting a thermal model of the case to the analysis, the temperature during analysis can be monitored at the Tj pin.

(1) Terminal configuration

The G3 model consists of five terminals



Port	Description
10	Drain
20	Gate
30	Source
50	Junction Temperature
55	Case Temperature

Figure 2-5 Terminal configuration (G3 model)

(2) Instance parameters

The following instance parameters are available for users to change:

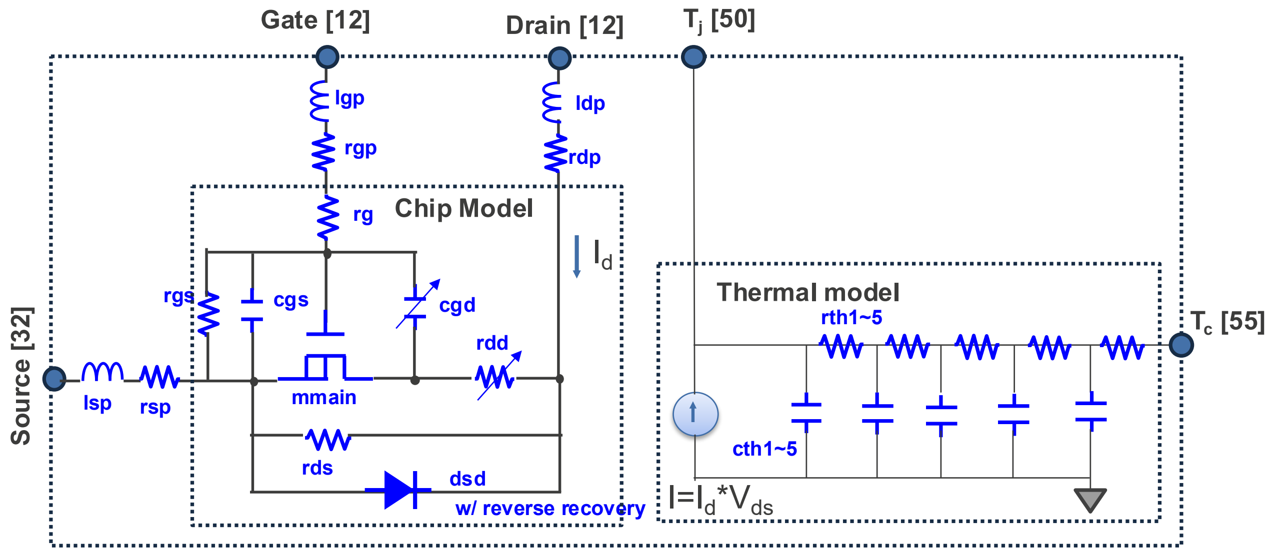
modsh: Selection flag for self-heating calculation. 0 disables the self-heating model.
tjmax: Upper limit for junction temperature calculation.
ggmin: Minimum conductance. Valid only for the target element. Used when convergence is poor.

Table 2-5 Instance parameters (G3 model)

Params	Default	Description
modsh	1	Self-heating off[0], on[1]
tjmax	500	Upper limit of Junction temperature calculation
ggmin	1e-15	Minimum conductance

(3) Model

G3 SPICE model



Parts	model
mmain	BSIM3 (PSpice Level=7, LTspice Level=8)
rg	Constant R
rdd	Constant R
cgd	Voltage variable C
dsd	Diode macro model w/ reverse recovery
rdp,ldp	PKG parasitics
rgp,lgp	PKG parasitics
rsp,lsp	PKG parasitics
rth1~5	Thermal R
cth1~5	Thermal C

Figure 2-6 Model equivalent circuit (G3 model)

3. How to use the thermal model(G3)

In addition to Drain, Gate, and Source, the G3 model has two extra thermal terminals. The Tj terminal is the temperature of the junction, and this terminal voltage is calculated as a temperature within the model (1V=1°C.) The voltage at the Tj terminal changes dynamically during the analysis. Therefore, this terminal should normally only be used to monitor the voltage, and nothing should be connected to it. By monitoring this voltage, the dynamic junction temperature during the analysis can be checked. The Tcase terminal is the outer shell or heat spreader portion of the power MOSFET package. Heat from the power MOSFET is transferred to the outside through this terminal

Figure 3-1 shows an example of the connection when the outer shell of power MOSFET is not in contact with anything, i.e., at ambient temperature. Tj should be open, and Tcase specifies the ambient temperature at the voltage source.

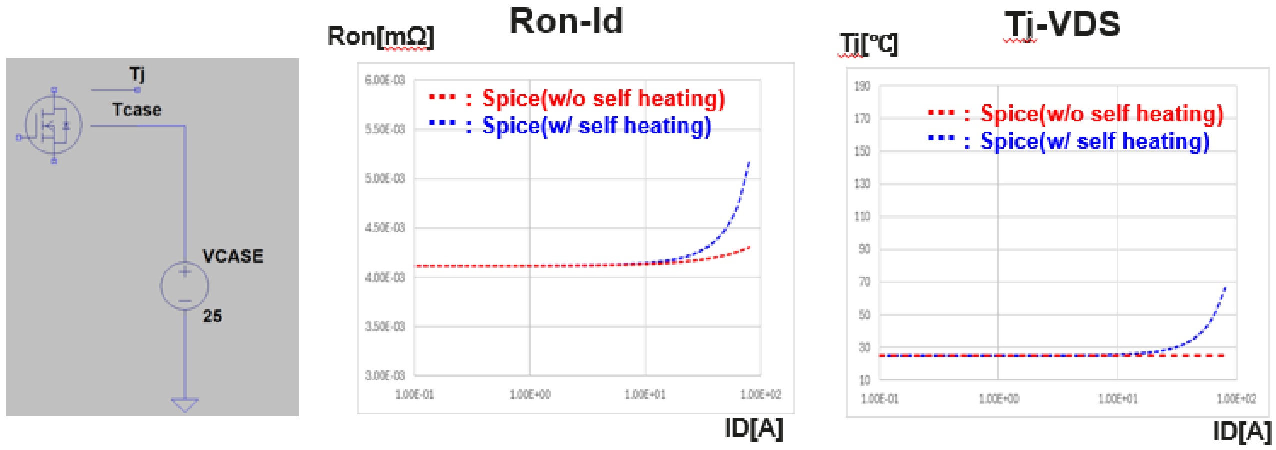


Figure 3-1 How to connect thermal terminals(a)

Figure 3-2 shows an example connection when power MOSFET is connected to a heat sink or similar device. Tj is set to open; Tcase connects a thermal model such as a heat sink, and a voltage source is connected to its outermost shell to specify the ambient temperature.

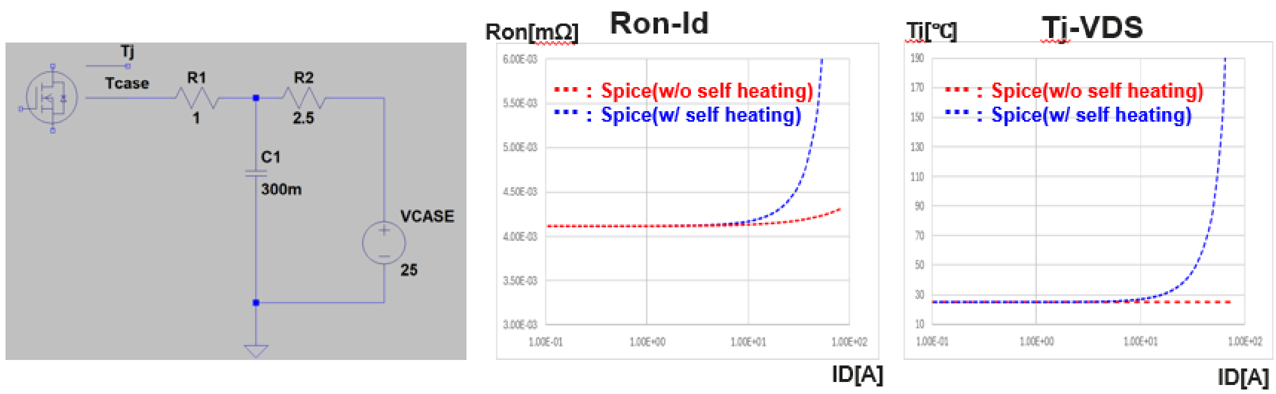


Figure 3-2 How to connect thermal terminals(b)

Figure 3-3 is a special case. If a voltage source is connected not only to the Tcase but also to Tj and the ambient temperature is specified, the Tj pin voltage is fixed at 25°C, which enables verification without the self-heating effect.

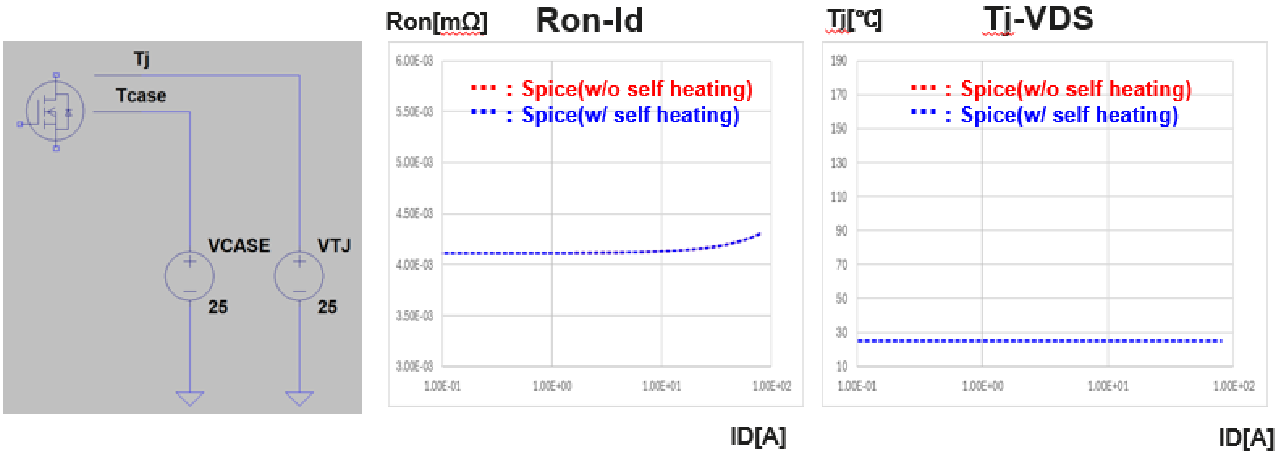


Figure 3-3 How to connect thermal terminals(c)

4. Accuracy Comparison

4.1 G1 vs G2 model

The following graphs show the comparison results between the measured data (red line) and the two SPICE models (G1 model: blue dotted line, G2 model: green dotted line).

Regarding the ID-VDS characteristics, it can be seen that the G2 model (green) has high fitting accuracy in all bias conditions. On the other hand, the G1 model shows a noticeable deviation from the VDS change. For on-resistance, the G2 model (green) also agrees well with the measured data (red) over a wide VGS range, whereas the G1 model (blue) deviates significantly, especially at low VGS, resulting in an overestimate of on-resistance. These results indicate that the G2 model reproduces the measured data with higher accuracy than the G1 model.

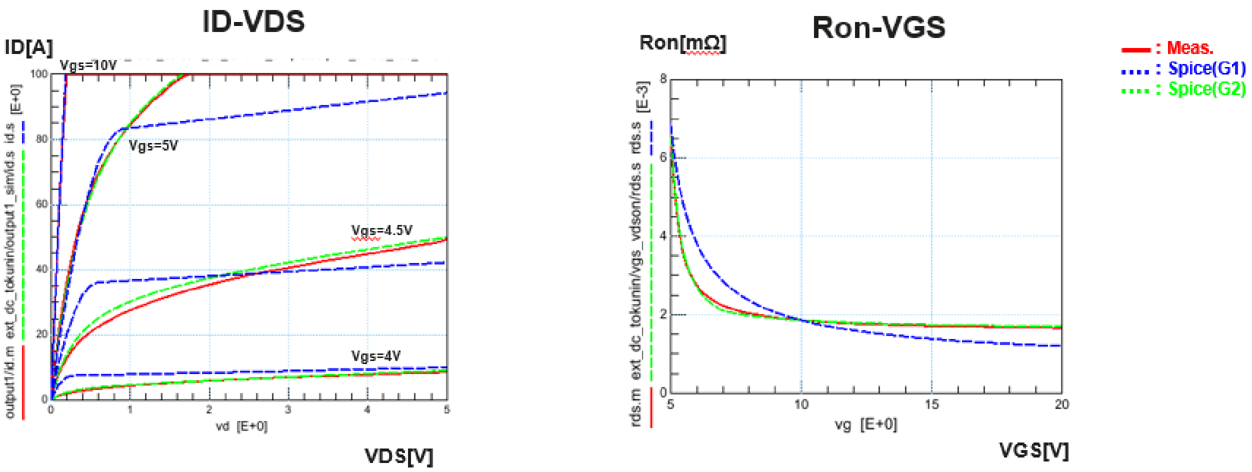


Figure 4-1 Comparison of modeling accuracy of DC characteristics (G1 vs. G2 model)

Power MOSFETs device structures become more complex with each generation, and the capacitance shows a complex bias dependence. The solid lines in the figure show the VDS dependence of Crss, Ciss, and Coss. On the other hand, the G1 model shows a conspicuous deviation from the actual measurements.

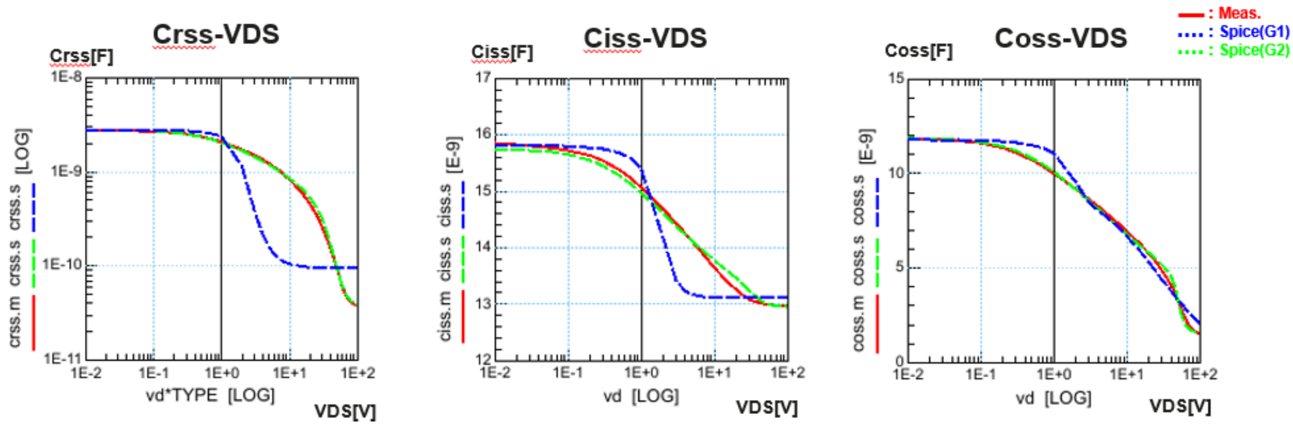


Figure 4-2 Comparison of modeling accuracy of DC characteristics (G1 vs. G2 model)

4.2 G2 vs G3 model

Figure 4-3 shows the results of the self-heating effect on the G3 model, which indicates that the ID tends to decrease as the temperature dynamically increases with VDS rise. In addition, since maximum junction temperature is set 500 ℃ as an instance parameter of G3 model, the maximum value of Tj is clipped at 500°C, and it can be confirmed that ID begins to increase thereafter.

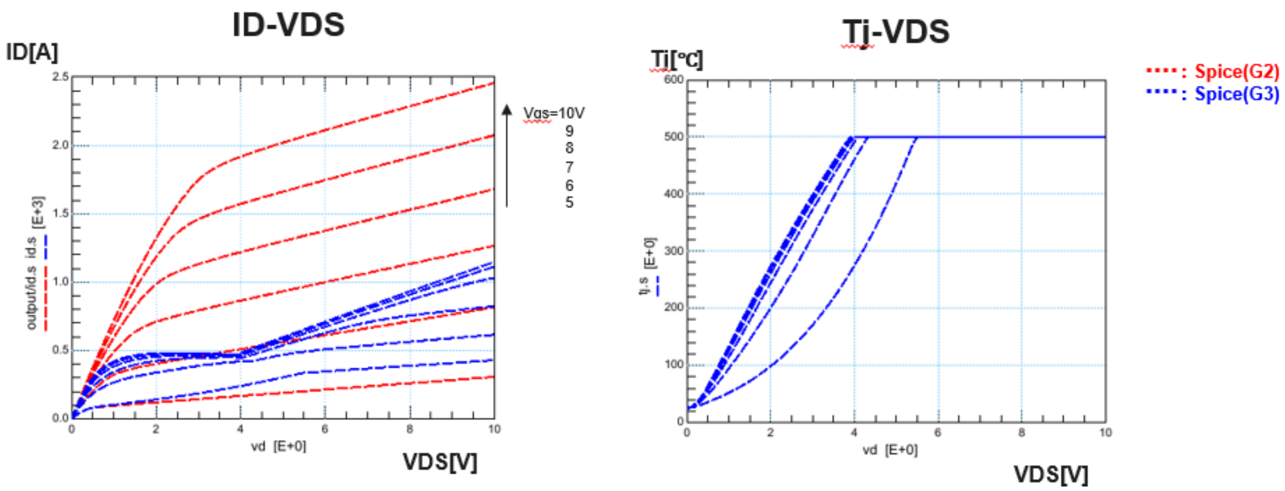


Figure 4-3 Comparison of modeling accuracy of CV characteristics (G1 vs. G2 model)

Figure 4-4 shows the results of verifying the self-heating effect of the On resistance in the G3 model, which shows a trend of increasing Ron due to the dynamic increase in temperature as ID rises.

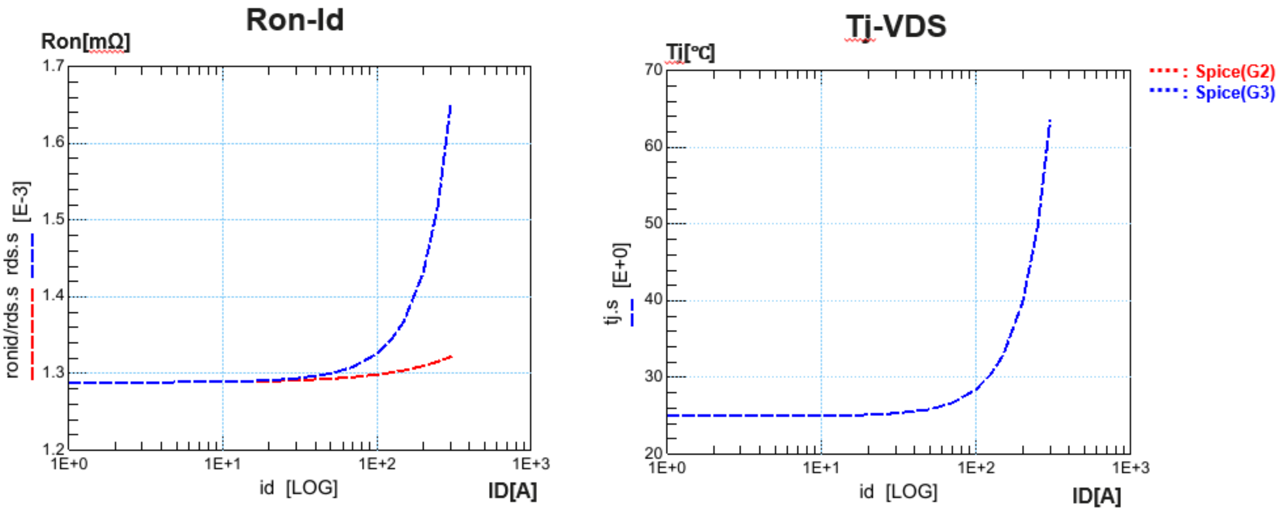


Figure 4-4 Comparison of modeling accuracy of Self-heating effect of Ron (G2 vs. G3 model)

5. Conclusion

This application note describes the three types of SPICE models provided by Renesas.

G1 model, which emphasizes calculation speed, G2 model, which emphasizes accuracy, and the G3 model, which can analyze the effects of self-heating effects. By selecting three SPICE models (G1/G2/G3) with consideration of their accuracy and calculation speed, users can choose the best simulation model and perform them for each purpose properly.

Revision History

Rev.	Date	Description	
		Page	Summary
1.0	May. 13. 2025	-	First edition

Corporate Headquarters

TOYOSU FORESIA, 3-2-24 Toyosu,
Koto-ku, Tokyo 135-0061, Japan
www.renesas.com

Trademarks

Contact information

For further information on a product, technology, the most up-to-date version of a document, or your nearest sales office, please visit: www.renesas.com/contact/.