

SPICE Model

Rev. 01



Nowadays, most users usually evaluate device performance through simulation to shorten the evaluation time. SPICE model of our GaN device is provided for such purpose. This document describes the equivalent circuit for GaN device and the flow for importing the model into SIMETRIX/SIMPLIS.

Equivalent circuit for GaN HEMT

Figure 1(a) displays that the simplified equivalent circuit for E-mode GaN device. It contains diodes for G-D and G-S side separately to model the gate leakage current. The Cgd, Cgs, and Cds are the parasitic capacitance which varied with voltage. In addition, it would have also parasitics contributed from the package. Figure 1(b) shows that the circuit for the GaN device contains parasitic resistances and inductances.





Steps to import to SIMPLIS/SIMetrix and simulate steps

This section shows briefly on how to import the model into SIMPLIS/SIMETRIX. The steps of importing model (.lib) file are described as follows.

Step 1 : Import the model file

- 1. Drag the model file (XXXX.lib) to the SIMetrix **command shell window**. (as shown in Figure 2)
- 2. Then, there will be a pop-up window asking you to confirm whether to install the model. -> Click **OK**.

le View	▼X Welcome		
Add Directory I-Modeling examples	SIMetrix / SII Pro with DVM - Version	MPUS on: 8.10b	
SIMetrix	Recent Schematics	D	ocumentation
	🧭 Open	6	What's New in Version 8.1?
	Z0200903_D70(MT95A)_v	wirebond_2 bank-3x5finger_Jill.sxsc 🦉	SIMetrix Documentation
	Z 20191120_D70_flip chip_2	2 bank-18finger_JJ.sxsch	SIMetrix User Manual
	Z0200727_D70(MV26)_w	rirebond_1 bank-18+2finger_JJ.sxsc - 🦉	Simulator Reference Manual
	20200727_D70(MV26)_w	rirebond_2 bank-18+2finger_JJ.sxsc - 🦿	Script Reference Manual
	20200903_D70(MT95A)_v	wirebond_1 bank-3x5finger_Jill.sxsc 🥑	Verilog A Manual
mmand Shell	20200903_D70(MT95A)_v	wirebond_1 bank-3x10finger_Jill.sx	
~	Ok Create New	D	esign Verification Module
	Schematic		DVM Tutorial
elcome to SIMetrix/SIMPLIS Pr /M arsion 8 10b (x64)	o with	6	DVM Documentation
** Warning ***: Another insta Metrix/SIMPLIS appears to b chematic auto-save will be d or this instance	nce of running. abled		
canning directory tree o cancel.	Pole-Zero Laplace Filter	Entry	Laplace - 3rd Order
del library changed. Rebuild	Parts Selector: Analog	Functions ►	IN OUT
ntalogs, please wait mpleted	複製 Laplace Transfer Fund	ctions >	1 real & 2 complex zeros 3 real poles
K	Nth-order Laplace Tra	ansfer Function	RTN
	Default Simulator		Show Welcome Page on Startup

Figure 2



Step 2 : Associate the model and symbol

- 1. Open a new schematic sheet (as shown in Figure 3)
- 2. Click Place -> From Model Library (as shown in Figure 4)





🖌 SIMetrix/SIMPLIS Main Window			
File Edit View Simulator	Place Probe Probe AC/N	Noise Hierarchy Monte Carlo Verilo	
🗋 - 📄 🗙 🖬 🕼 🖲	Repeat Last Place	Alt+R 🛛 🔍 🤤 🕀 🔍 🧷	
File View	2. From Model Library	Ctrl+G X untitled	
Add Directory	From Symbol Library Annotation Select by Specification Search Part Hierarchy Create Model Magnetics Passives		

Figure 4



- 3. Select the "Recently Added Models" on the left window (as shown in Figure 5)
- 4. On the right window, you would find the model name you installed.
- 5. Select the model and press "Place"

🖌 Select Device	×
* Recently Added Models *	A_Sample
* All User Models *	
* All Models *	
Analog switches	
BJT - Dual	
BJT - Multiple	
Bridge Rectifiers	
Buffers	
Comparators	
Counters	
Current Mirror	
Content Minors	
SIMetrix does	
not know what	
symbol to use	
for this model	
Drock "Diaco"	
Fless Flace	
to resolve	
	ς Ι
	J.
Eller 8	Plan Canal Hale
глят	Place Cancel Help

Figure 5

6. If SIMetrix doesn't know what symbol to use for the model, you will see a window as below. (as shown in Figure 6)

🖌 Associate Symbol with Model A_Samp	le		? ×	
STEP 1. Select a suitable category for this part. If you can't find one, press "New Category" and enter a new category of your choice. STEP 2. Select a suitable symbol for this part. Select a symbol from the drop down box or press "Auto Create Symbol" to create a new one. If selecting an existing symbol you must make sure that the pin order matches the model definition shown below. Press "Help" for assistance.	Choose Category for A_Sample			
	Unassigned		▼ New Category	
	Define Symbol for A_Sample			
	Amplifier (subcircuit)		Auto Create Symbol	
	Pin order			
		••••••••••••••		
If you use "Auto Create Symbol" the pin order will not need to be changed.		·····	· · · · · · · · · · · · · · · · · · ·	
		••••••	· · · + · · · · · + · · ·	
		•••••••••••••		
	Edit Pin Names		· · · · · · · · · · · · · · · · · · ·	
Electrical Model - A Sample		han an a		
			^	
			*	
		Ok Ca	ncel Help	

Figure 6



- 7. You can choose a built-in symbol (NMOS 3 terminal for 3 pins / N-channel 4 terminal_KS for 4 pins , Figure
 7) or follow the steps below to create a new symbol.
 - I. First, you have to select the category for the model. If you can't find a suitable category, you can create a new one by pressing "**New Category**".
 - II. Next, please select a symbol for the model. If you can't find a suitable symbol, you can press "Auto Create Symbol" to create a new one. If the pin names aren't suitable, you can define the pin names by pressing "Edit Pin Names".
 - III. Click **OK** to finish the symbol definition of the model.

Define Symbol for A_Sample		
NMOS 3 terminal		
Pin order	▲ ↓	
D		
G		: :
s	· · · · · · · · · · · · · · · · · · ·	: :
		• •
		: :
		: :
		• •
Edit Pin Names		



🖌 Associate Symbol with Model A_Sampl	e	? ×
STEP 1. Select a suitable category for this part. If you can't find one, press "New Category" and enter a new category of your choice.	Choose Category for A_Sample Unassigned	▼ New Category
STEP 2. Select a suitable symbol for this part. Select a symbol from the drop down box or press "Auto Create Symbol" to create a new one.	Define Symbol for A_Sample Amplifier (subcircuit)	Auto Create Symbol
If selecting an existing symbol you must make sure that the pin order matches the model definition shown below. Press "Help" for assistance.	Pin order	
If you use "Auto Create Symbol" the pin order will not need to be changed.		
	Edit Pin Names	
Electrical Model - A_Sample		^
	Ok	ancel Help



Simulate and compare to experimental results

Model 600_70m_MT95A_P85C is used to compare on the device characteristic with measured data. In the below simulation, Vgs from 0V to 6V, and the Vds set at 0.05V are used. It shows that the IdVd & IdVg curve output in Figure 9. The solid line is measured data by Agilent B1505A and the dashed line is the simulation result.



Figure 9(a) IdVd curve



9(b) IdVg curve





Figure 10 shows the curve for device capacitances (Crss/Ciss/Coss) which varied with D-S voltage from 0V to 600V. The data is measured by Agilent B1505A as well.



Figure 10

Then, in some case even when GaN is turned off, the current may flow in the reverse direction. In the reverse condition, Vsd is biased by Vth - Vgs and increases with the reverse current. Figure 11 shows the Vsd curve with Vgs=-5V~0V.



Figure 11



Revision History

Revision	Date	Description of Change
00	2020-12-23	First Release
01	2021-08-18	Added how to choose build-in symbol