ANCORA
碇基

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.

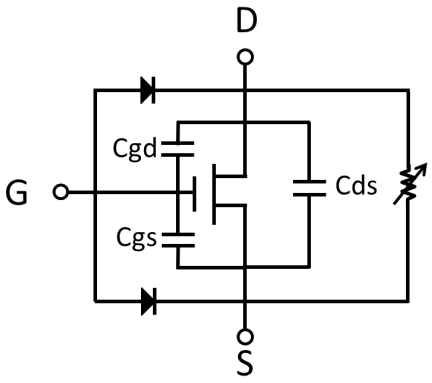
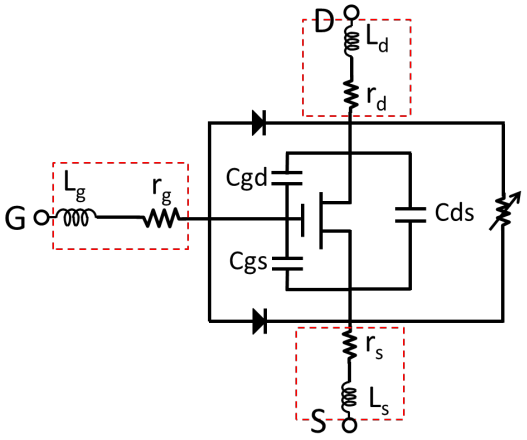


Figure 1(a)



1(b)

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.**

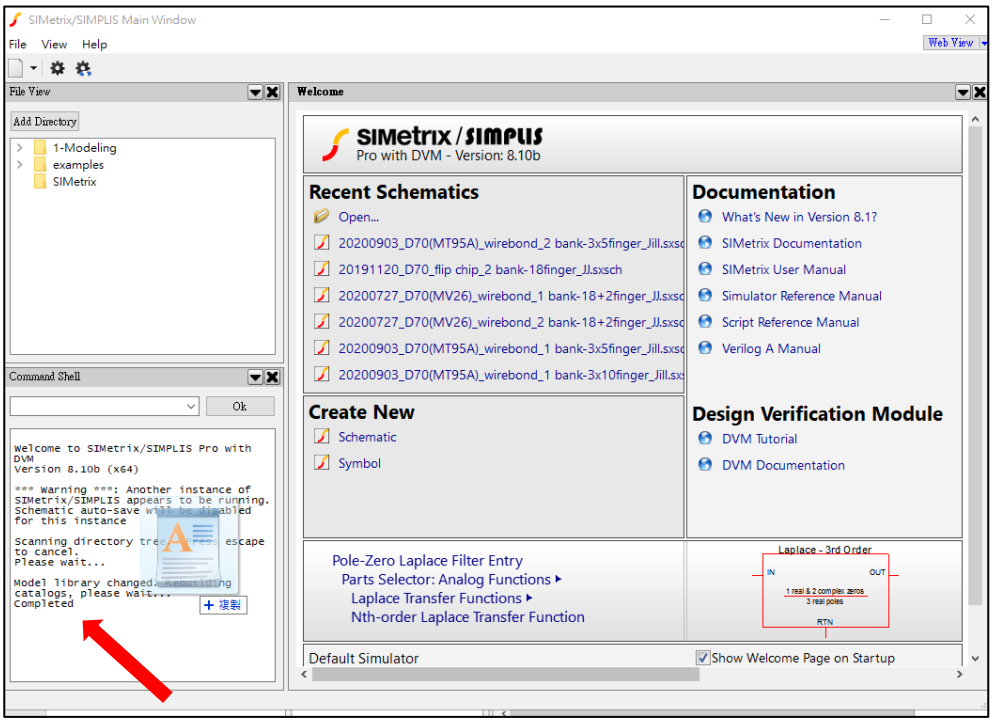


Figure 2

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)

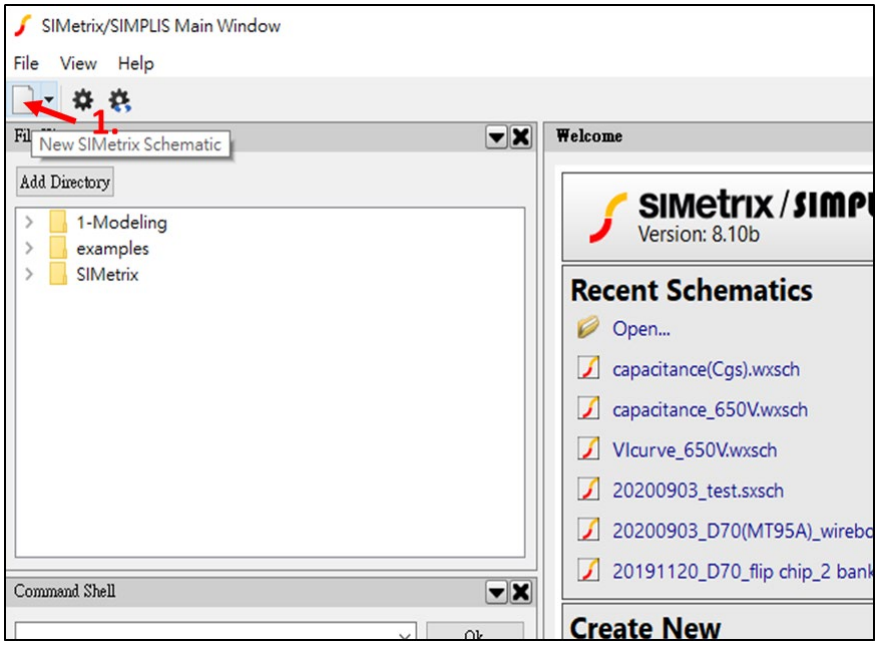


Figure 3

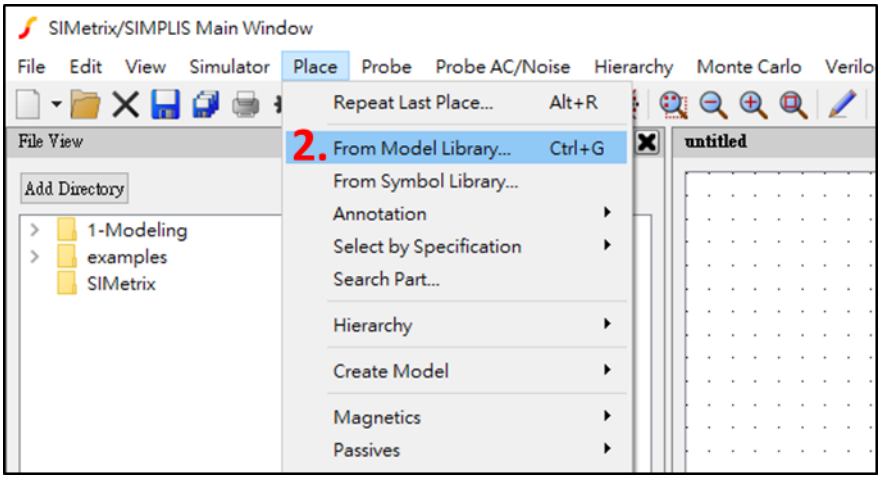


Figure 4

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**"

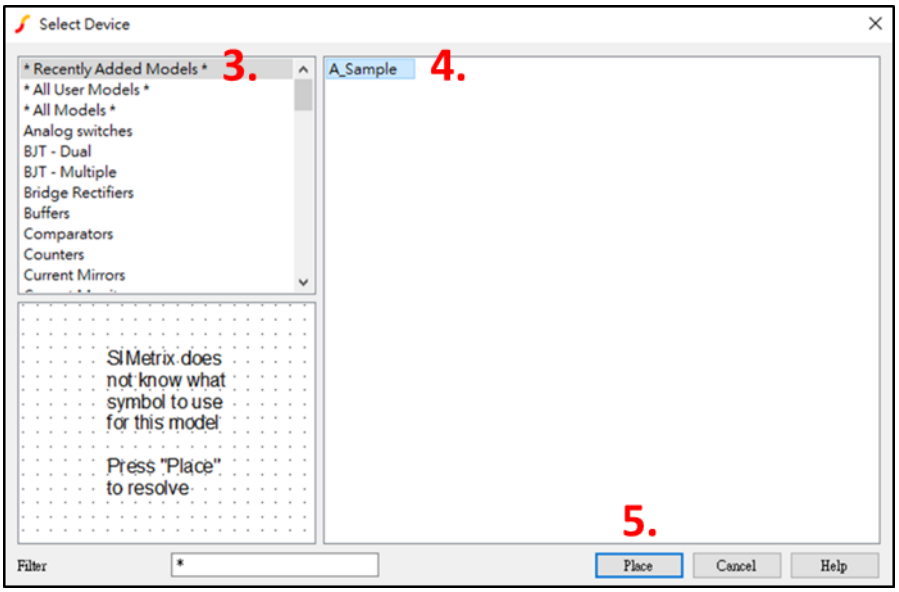


Figure 5

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)

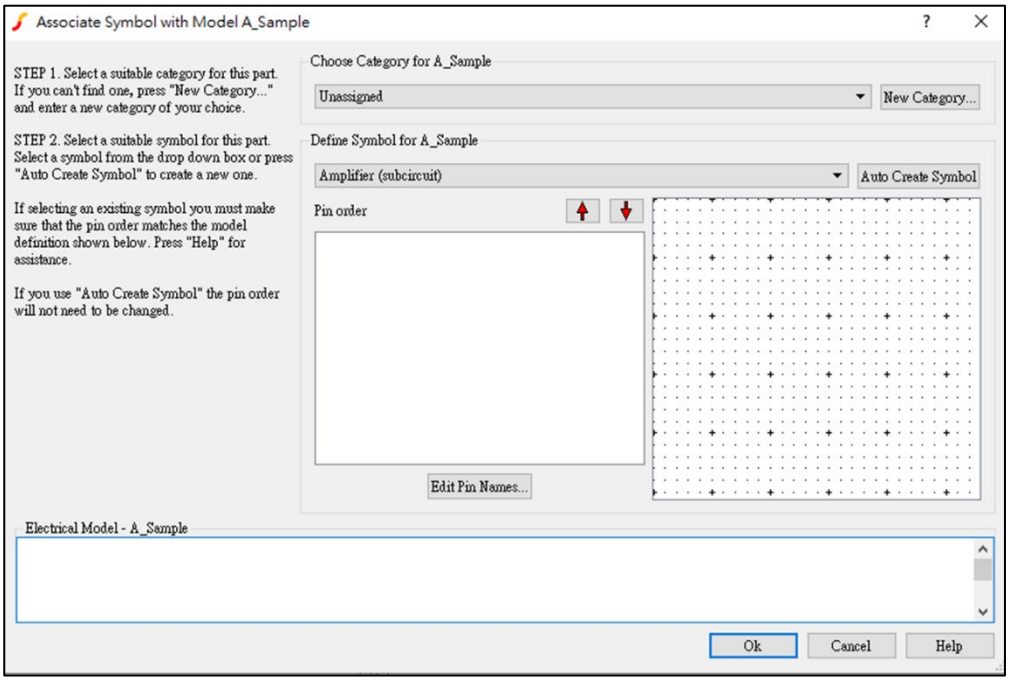


Figure 6

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.

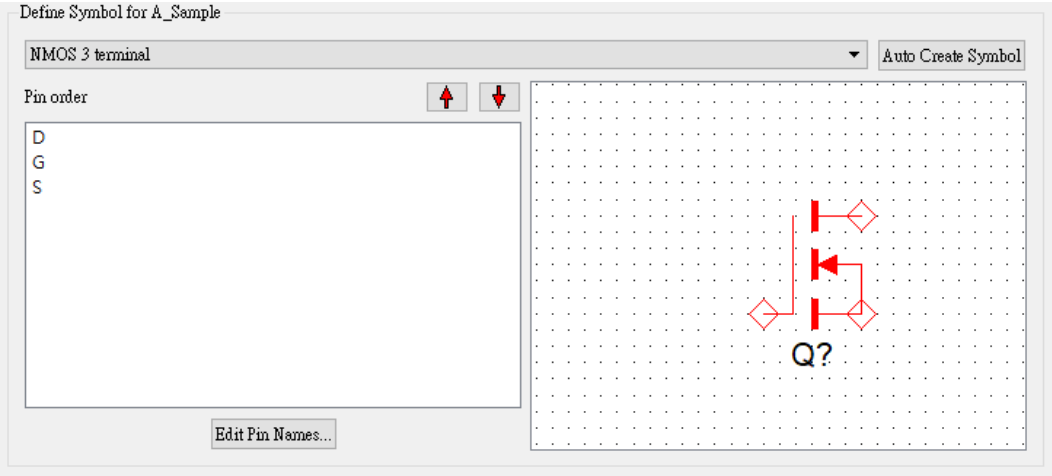


Figure 7

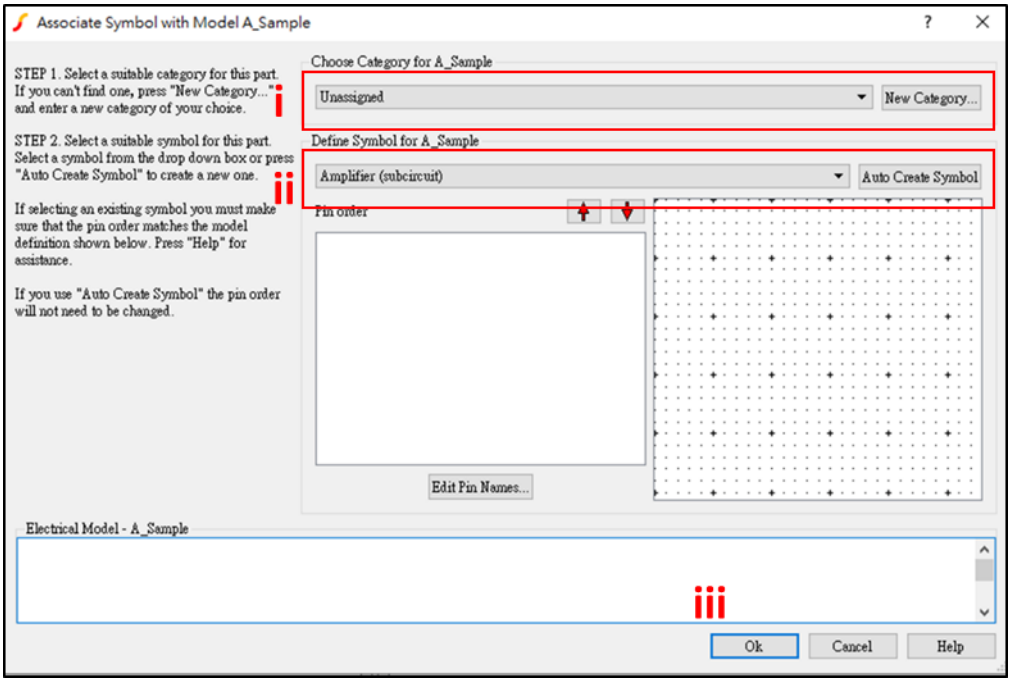


Figure 8

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.

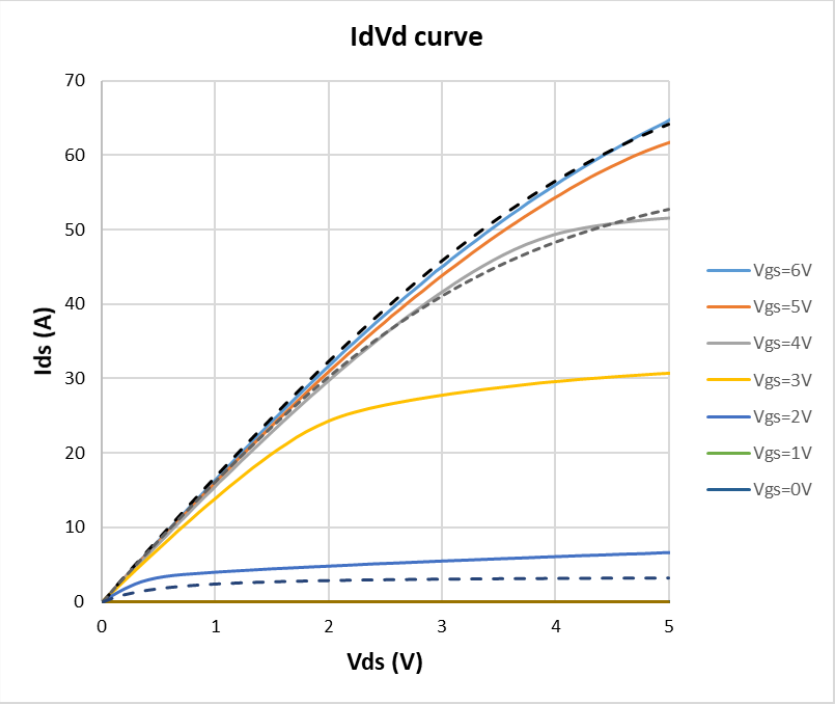
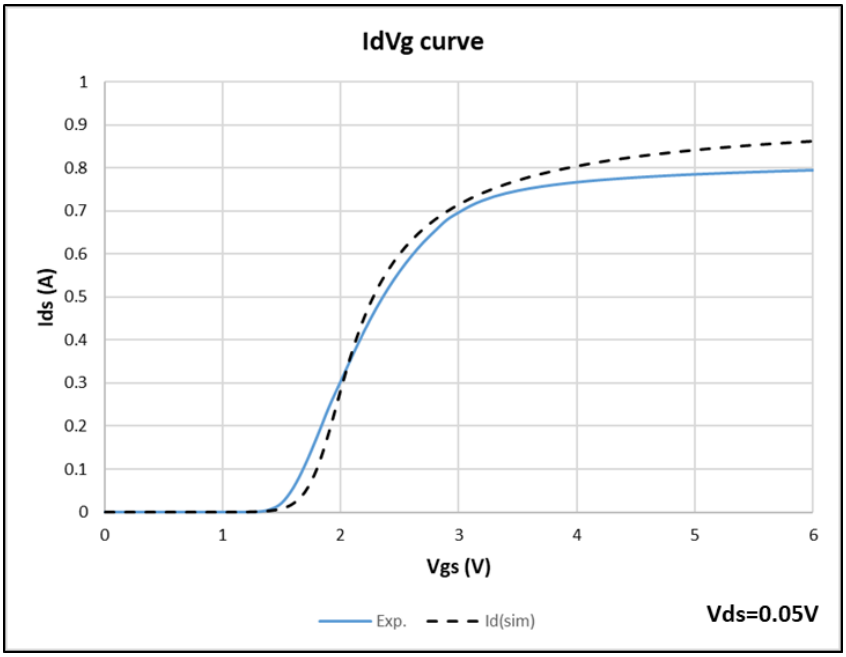


Figure 9(a) IdVd curve



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.

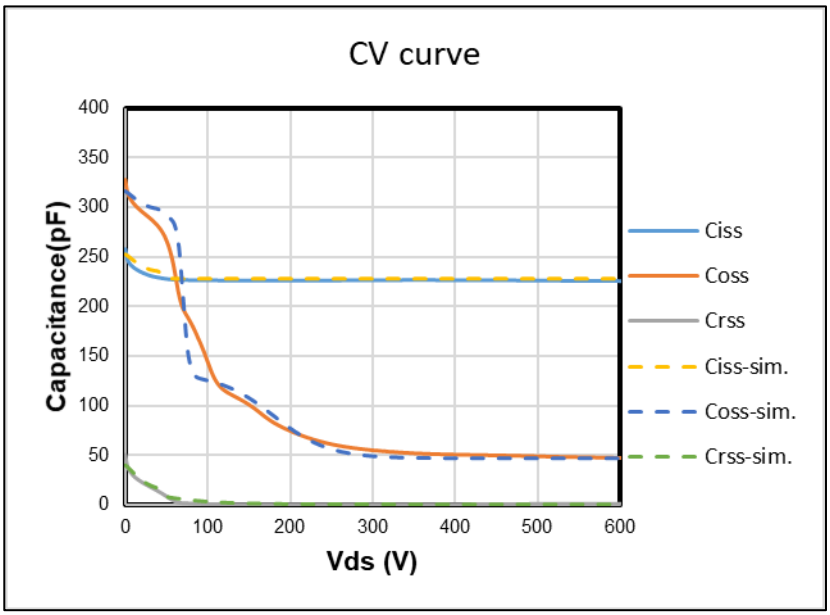


Figure 10

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.

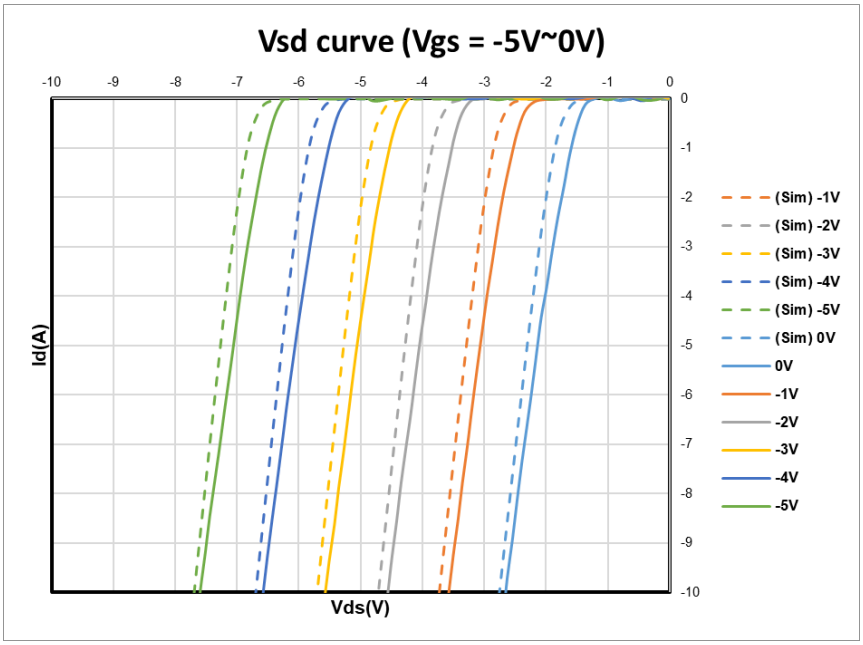


Figure 11

Revision History

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