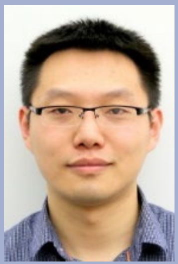
APPLICATION NOTE
UnitedSiC_AN0005 – August 2019

Using UnitedSiC SPICE model in LTSPICE

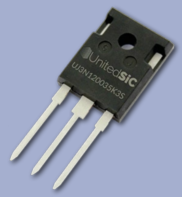
Zhongda Li, Ph.D.

Introduction

This application note explains how to use UnitedSiC SPICE models in the popular circuit simulator LTSPICE.



Zhongda is a Senior Staff R&D Engineer at United Silicon Carbide. Experience includes SiC device designs, product development, and generating SPICE models.



Learn more about power electronic applications at https://unitedsic.com/downloads

Downloading the UnitedSiC SPICE Models

New	Part Number	Package	V	RDS(on) typ	Id max	Data Sheet	SPICE Model	Buy Now	Samples
	Part Number								
	UF3C065030K4S	TO-247-4L	650 V	30 mΩ	85.0 A	Data Sheet	SPICE	Order Online	
	UF3C065080K4S	TO-247-4L	650 V	80 mΩ	31.0 A	Data Sheet	SPICE	Order Online	
	UF3C120040K4S	TO-247-4L	1200 V	35 mΩ	65.0 A	Data Sheet	SPICE	Order Online	
	UF3C120080K4S	TO-247-4L	1200 V	80 mΩ	33.0 A	Data Sheet	SPICE	Order Online	

Figure 1 UnitedSiC FET Selector Web Page (https://unitedsic.com/cascodes/)

The SPICE model file is in text format (*.txt file). Simply click the "SPICE" icon to open the SPICE model file (text, ".txt) in the browser, as shown in Figure 1, and copy and paste into Notepad and name the file with the part number. For example, for the part UF3C120080K4S, the SPICE file name should be "UF3C120080K4S.txt".

Note that you do not necessarily have to make the file extension to be ".txt". The file extension can to be anything, such as "lib" or "sub", and the model will still work.

Downloading the UnitedSiC Symbol Files (*.asy) for LTSPICE

UnitedSiC_Diode.asy
UnitedSiC_FET_3L.asy
UnitedSiC_FET_4L.asy
UnitedSiC_JFET_3L.asy

Figure 2 UnitedSiC Symbol Files (*.asy) for LTSPICE

Next download the symbol files as shown in Figure 2 and save them under the LTSPICE default symbol folder. The folder is by default under:
C:\Users*YOUR_USER_NAME*\Documents\LTspiceXVII\lib\sym

We recommend creating the "UnitedSiC" folder and copy the four .asy files into it.

Configuration of LTSPICE

```
.options ABSTOL=1n
.options CHGTOL = 1p
.options ITL1= 150
.options ITL2 = 150
.options ITL4 = 500
.options RELTOL =0.001
```

Figure 3 Content of the "SPICE Options.txt" File

The "SPICE Options.txt" file contains the options to set numerical accuracy of the SPICE simulator and can be downloaded from UnitedSiC website. We recommend the values shown in Figure 3 based on the optimized trade-off between simulation accuracy and ease of convergence.

Once this file is downloaded, you can use the SPICE command ".inc" to include the file in your circuit file. We will show how to do this in later section.

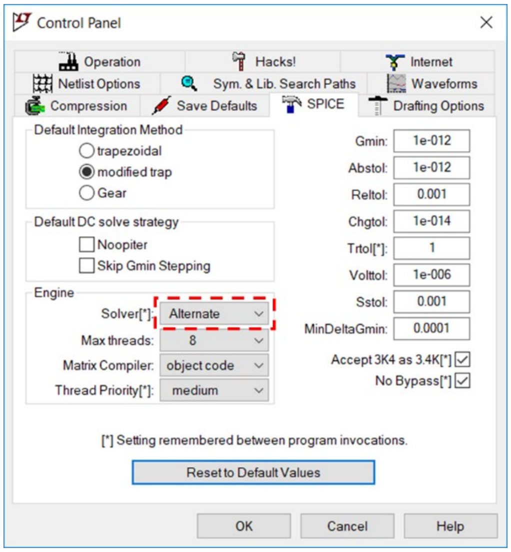


Figure 4 LTSPICE Control Panel Solver Setting

The last step is in the LTSPICE menu open "Tools -> Control Panel", and under the "SPICE" tab, change the solver to "Alternate" from "Normal". Per LTSPICE manual, the "Alternate" solver runs at half the speed but with one thousand times more internal accuracy. We recommend this setting to enhance convergence and accuracy when simulating UnitedSiC devices.

Using the UnitedSiC Models for SPICE Simulations

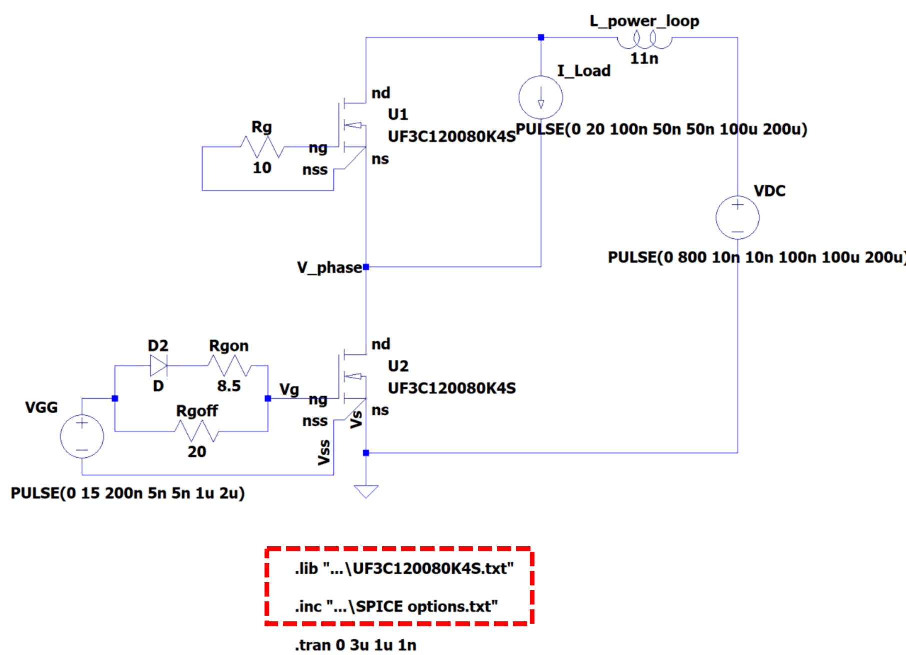


Figure 5 Example Half-bridge Double Pulse Circuit

Now let us use a simple half-bridge double pulse circuit to demonstrate how to build and simulate a circuit, as shown in Figure 5.

First type the ".lib" command to point to the location where the UnitedSiC SPICE model file is saved. And type the ".inc" command and point to the "SPICE Options.txt" file. See Figure 5.

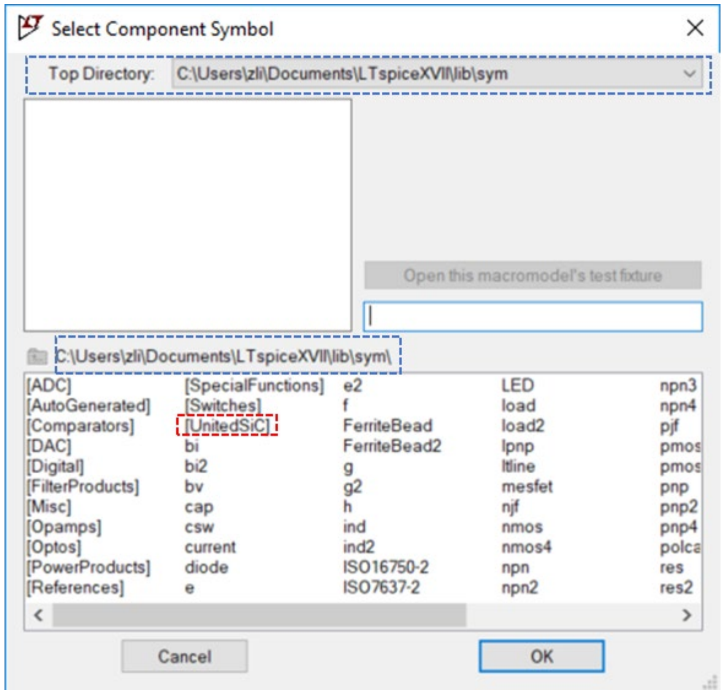


Figure 6 LTSPICE "Select Component Symbol" Window

Now let us insert the UnitedSiC UF3C120080K4S device into the circuit. From menu, go "Edit -> Component". This should open the "Select Component Symbol" Window, as shown in Figure 6. The default directory you see should be
C:\Users*YOUR_USER_NAME*\Documents\LTspiceXVII\lib\sym
And you can locate the UnitedSiC symbol files in the UnitedSiC folder, if you have created one.

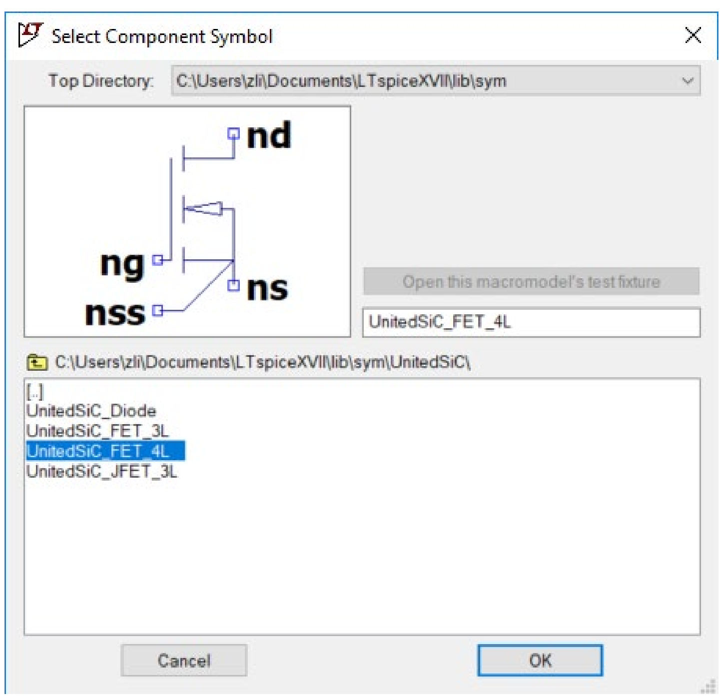


Figure 7 Selecting the UnitedSiC Symbol

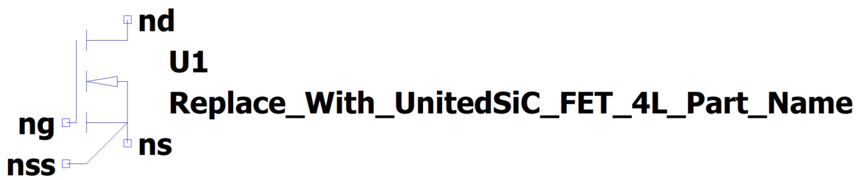


Figure 8 UnitedSiC SiC FET Symbol

Because in this example the UF3C120080K4S part is SiC FET in 4-lead TO247 package, we choose the symbol "UnitedSiC_FET_4L", as shown in Figure 7. The inserted symbol looks like Figure 8.

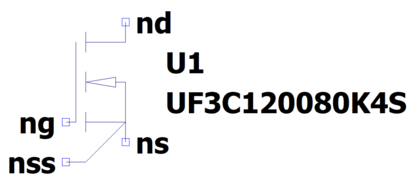


Figure 9 UnitedSiC SiC FET Symbol After Replacing With Correct Part Name

Now right click the text "Replace_with_UnitedSiC_FET_4L_Part_Name", and type in the part name you would like to use.

After inserting the other circuit components and setting up the transient simulation, we can run the simulation and probe the Vgs, Ids, and Vds of the lower FET:

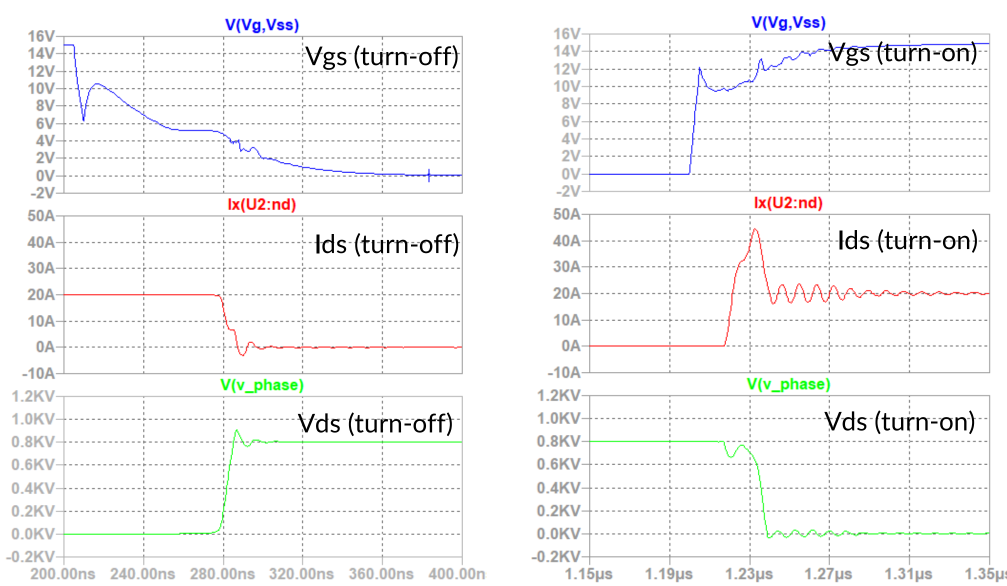


Figure 10 Switching Waveforms from LTSPICE Transient Simulations