How to use Physical and Scalable Models with SIMetrix, OrCAD and LTSpice

INTRODUCTION

ON Semiconductor has developed Spice Physical and Scalable SPICE simulation models based on the device silicon structure. The models are available for discrete components under the dedicate component landing page.

The advanced models are described in several papers, please see the "SPICE Modeling Tutorial" [1] at ON Semiconductor web site in the "Design Support > Technical Support > Tutorial" section.

Direct link: http://www.onsemi.com/pub/Collateral/TND6248-D.PDF

The objective of the paper/tutorial is to explain how to implement the models in the various simulators. ON Semiconductor models are available on the web for SIMetrix/SIMPLIS [2], OrCAD PSpice® [3] and LTSpice® [4]. Other simulators can be supported upon request. This tutorial explains how to add the simulation models to the library and how to run a simple simulation example with the model and symbol libraries.

This tutorial assumes the simulation software is already installed and, also, basic knowledge regarding the simulator (how to capture the schematic, make and run a basic simulation schematic).

This tutorial goes step by step from ON Semiconductor web page to find models online down to the point where the simulation results are obtained. To do that, the training is based on a SuperFET 3 device (super junction MOSFET): FCD360N65S3R0 as an example to measure the ON drain resistance at 1 A drain current and with 10 V gate voltage. The FCD360N65S3R0 online product page can be found at: <u>http://www.onsemi.com/PowerSolutions/product.do?id=FCD360N65S3R0</u>



ON Semiconductor®

www.onsemi.com

APPLICATION NOTE

ON Semiconductor®



Document type: Product Taxonomy: Application:

- or - Enter Part#/Keyword Searc

All Simulation Models
1 - 50 of 15622 [1 2 3 4 5 6 7 8 9 10 next [>>]]

Document Title	Document ID/Size
650V SIC JBS Gen1.5(LTSPICE) NEW	ONSEMI_SICDIODE_650V_GEN1P5_REV1(LTspice Model) (60kB)
650V SIC JBS Gen1.5(SIMETRIX) NEW	ONSEMI_SICDIODE_650V_GEN1P5_REV1(SIMetrix Model) (19kB)
650V SIC JBS Gen1.5(PSPICE) NEW	ONSEMI_SICDIODE_650V_GEN1P5_REV1(PSpice Model) (16kB)
PTNG_100V_PSPICE NEW	PTNG_100V_PSPICE (50kB)
PTNG_100V_Simetrix NEW	PTNG_100V_Simetrix (57kB)
PTNG 100V LTSPICE NEW	PTNG 100V LTSPICE (216kB)

1

Table of Contents		Table	of	Contents	
-------------------	--	-------	----	----------	--

Introduction 1
Obtaining the models
From the product page
From the model landing page
From the main search box
Simulation setup with SIMetrix
Preparing files for SIMetrix
Dragging the sub-circuits' file to SIMetrix
Dragging the symbols' file to SIMetrix
Dragging both files
Adding the sub-circuits' file using SIMetrix menu
Adding the symbols' file using SIMetrix menu
Simulation setup with Orcad PSpice
Adding file to OrCAD PSpice library directory
Setting the library as local for one project
Creating a blank PSpice project with the symbols for the schematic capture
Creating a blank Simulation Profile with the sub-circuits for simulation
Setting the library files as global for all projects
Setting symbols for the schematic capture environment
Setting sub-circuits for PSpice simulation environment
Simulation setup with LTspice
Adding files to LTSpice library directory
Special access to LTSpice library folder on the MAC version
Setting the library in LTSpice 24
Creating a blank schematic with the library statement
Editing symbols to link them to the library 24
Fast symbols' files editing with a text editor
Simulate with ON physical and scalable component models
Schematic capture
With SIMetrix
With OrCAD
With LTSpice
Simulation setup
With SIMetrix
With OrCAD
With LTSpice
Results
With SIMetrix
With OrCAD
With LTSpice
Conclusion
Biography

Table of Figures	
Figure 1. Online product page	4
Figure 2. Links to models' available from product page	4
Figure 3. ON Semiconductor main web page menu	5
Figure 4. Technical Support landing page	5
Figure 5. Simulation Models landing page	6
Figure 6. Simulation models search box	6
Figure 7. List of available models after searching	6
Figure 8. Main search box	7
Figure 9. Main search box results page	7
Figure 10. Unzipping the library files	8
Figure 11. Importing the sub-circuits' text file	8
Figure 12. Importing sub-circuits validation	9
Figure 13. Importing the symbols' file	9
Figure 14. Importing symbols validation	9
Figure 15 Importing the sub-circuits' text file and the symbols file together	0
Figure 16. Selecting sub-circuits' text file added	0
Figure 17. Sub-circuits' library waiting to be included 1	1
Figure 18. Library selection and inclusion	1
Figure 19. Sub-circuits' library text file added 1	1
Figure 20. Symbol Manager to include the symbols' library	2
Figure 21. Selecting symbols' file	3
Figure 22. Symbols added in the symbols library	3
Figure 23. Unzziping the library files	4
Figure 24. Copying ON symbols library file in OrCAD Capture library directory for PSpice	4
Figure 25. Copying ON sub-circuit library file in OrCAD Pspice library directory	5
Figure 26. New project window setup	5
Figure 27. New project setup	6
Figure 28. Complete project tree	6
Figure 29. ON symbol library loaded in the project	0
Figure 30. New simulation prome to include sub-circuits delimitons	0
Figure 31. Bids point simulation as default profile	7
Figure 32. Selecting the text file containing the sub circuite' definitions	/ 0
Figure 33. New simulation profile setup with sub-circuits' definitions included	o Q
Figure 35. Project tree with all ON library	a
Figure 36. New project window setup	9 0
Figure 37. New project setup	a
Figure 38 Add library button	0
Figure 39. Symbol library file added as global	0
Figure 40. New simulation profile setup with sub-circuits' definitons included	1
Figure 41 Unzigning the library files	2
Figure 42 All symbols available	2
Figure 43. Library path	2
Figure 44. ON sub-directories for sub-circuits' definitons included	3
Figure 45. Library folder access on a MAC	3
Figure 46. Library call simulation statement	4
Figure 47. TSpice symbol editor	4
Figure 48. Add the sub-circuits' definitions' file path to the symbol	5
Figure 49. LTSpice symbol text file example with the ModelFile statement	6
Figure 50. ON resistance measurement setup with SIMetrix	7
Figure 51. ON resistance measurement setup with OrCAD PSpice	7
Figure 52. ON resistance measurement setup with LTSpice	8
Figure 53. Simulation analysis selection with SIMetrix	8
Figure 54. Inherit simulation profile creation with OrCAD PSpice	8
Figure 55. New simulation command creation with LTSpice	9
Figure 56. Simulation results obtained with SIMetrix	9
Figure 57. Simulation results obtained with OrCAD PSpice	0
Figure 58. Simulation results obtained with LTSpice	0

OBTAINING THE MODELS

ON Semiconductor models are available on ON Semiconductor web site. There is several ways or paths to find the model corresponding to one device.

From the Product Page

From the main web page, use the search box or the menu to get to this product landing page. The FCD360N65S3R0 online product page should look like this:

ON Semiconductor® ON Semiconductor Construction Construct	tions 📕
Products Applications Design Support About MyON Search Sear	
Home > Products > Discrete > IGBTs & FETs > MOSFETs > FCD360N65S3R0 Previously Viewed Products Select Product	C
Select Product.	(<u>1</u>
FCD360N65S3R0: N-Channel SuperFET [™] III MOSFET, 650 V, 10 A, 360 mΩ	Go Clear List
Total Datasheet: N-Channel Total -View Material Composition Design Support	
SUPERFET® III Easy-Drive Compared and the set of the set	
Rev. 3 (248kB) Design Resources & Documents	<u>1</u>
SuperFET® III MOSFET is ON Semiconductor's brand-new high voltage super-junction (SJ) MOSFET family that is utilizing charge balance Technical Support	
technology tor outsitanding for automatic consultation of the second	
Features Benefits	
700 V @ T₁ = 150 °C • Higher system reliability at low temperature operation	
Ultra Low Gate Charge (Typ. Gg = 18 nC) Low Sate	
Low Einscure output capacitation (yp: costs(en.) = (r/s pr) Content sentencing loss Optimized Capacitance Covert sentencing loss Covert sentencing loss Covert sentencing loss	
Internal Gate Resistance: 1 Ω	
• Typ. Region = 310 mΩ	
Horis Conjunction Tested	
Applications End Products	
Computing Notebook / Desktop computer / Game console	
Consumer Consumer Consumer Industrial CD/ LED TV	
LED Lighting / Ballast	
Adapter	
Technical Documentation & Design Resources	
Application Notes (12) Data Sheets (1)	
Simulation Models (3) Package Drawings (1)	

Figure 1. Online Product Page

On the bottom, a "Simulation Model" link can be found with the number of models available. In this example, there are 3 models available.

Moving the mouse on the link displays the type of models available.



Figure 2. Links to Models Available from Product Page

Then:

- Click on "SIMetrix SPICE model ..." for SIMetrix simulator.
- Click on "PSPICE model and symbol ..." for OrCAD PSpice platform.
- Click on "LTspice model and symbol ..." for LTSpice simulator.

It will start downloading the zip file containing all models available for the complete SuperFet 3 family.

As explain in James Victory's tutorial [1], those models are developed for a particular technology with physical equations and not for one device by curve fitting.

From the model landing page

From the main web page, use the menu to go to "Design support > Technical Support"



Figure 3. ON Semiconductor Main Web Page Menu

By clicking, the following web page appears:

ON Se	emicondu	ctor®		Energy Efficient Innova	简体中文1日本語 Itions
Products	Applications	Design Support Design Support	MyON	Search 🔶 spice modeling tutoria	Q
Design Support Technical Support Blog Power Supply WebDesigner Interactive Block	Home > Support : Technical S ON Semiconducto	 Design Support > Technical Supp upport 	ort ons you may have.		Previously Viewed Products Select Product C Co Clear List
Diagrams	Frequently A	sked Questions		Email Technical Support	Documents
rToduct Recommendation Tools	Where Applications Taxonomy Product Functional Search	Question C contains Select Select	0	Send email to Technical Support Engineers about product, support on application issues. Most responses provided within 48 hours. >> Email Technical Support Sales & Distribution Support Find your local sales and distribution offices. Select a country: FRANCE © Go	 Application Notes Coltarent Brochures Conformance Reports Data Bhoetas Data Bhoetas Deaign Notes Selauten Read Decuments Selauten Read Podatas Podatas Development Tools Podatas Development Podot Podatas Development Recommendation Tools Pedratas Designs
	Phone Techr	ical Support			» Reference Manuals » Simulation Models
	Technical sup	port representatives are available at o	ur Technical Support Centers to a	nswer questions about our products.	» Videos » Simplis Models Software
	United State	s and the Americas			* White Papers
	Voice Mail	1 800 282 985	5		
	Phone	011 421 33 79	0 2910		
	Business Ho	urs Monday-Frida	y, 9:00AM - 5:00PM MST (GMT -0	17:00)	
	Europe, Mid	die East and Africa			
	Phone	00421 33 790	2910		
	Business Ho	urs Monday-Frida	v. 9:00AM - 5:30PM CET (GMT +	01:00)	

Figure 4. Technical Support Landing Page

On the right-hand side, there is a section named: "Design Resources & Documents". In this section find the "Simulation Models" link and click it.

See below the simulation models landing page that should look like a list as shown:

ON Semicon	ductor®	Energy Efficie	nt Innovations	简体中文1日本語
Products Application	s O Design Support O About	O MyON Search 🗢	spice modeling tutorial	A III
Design Support	Home > Support > Design Support > De	sign Resources & Documents > Simulation Models	Previously V	ewed Products
	Oliver Handala		Select Produ	ct 🔅 Go
	Document type: Product Taxonomy: Application: - or - Enter Part/Riceyword All Simulation Models 1 - 50 of 15600 [1 234567891	Search 0 next [>>]]	Technical S * Email Tech * Line Tech * Dia Tech * Find a Dist	Clear List upport inical Support ical Support hnical Support inclufor Page size: 60 0
	Document Title 🕶	Document ID/Size	Revision	Revision Date
	ESD8104MUTAG.LIB	ESDB104MUTAG.LIB (1.0kB)	0	
	IBIS Model for MC100EP016A_33V	MC100EP016A_33V.IBS (11kB)	2	Oct, 2016
	IBS Model for MC100EL11D 5.0V	MC100EL11D_PECL.IBS (16.0kB)	2	
	IBS Model for MC100EP16DT 3.3V	MC100EP16DT_33.IBS (5.0kB)	2	
	IBS Model for MC100EP31DT_33	MC100EP31DT_33.IBS (5.0kB)	2	
	IBS Model for MC10EP11D_50	MC10EP11D_50.IBS (5.0kB)	0	
	LIB Model for MJD44H11	MJD44H11.LIB (0.0kB)	0	
	LIB Model for MJD47	MJD47.LIB (0.0kB)	0	
	LIB Model for MMBT2482LT1	MMBT2482LT1 LIB (0.0kB)	0	

Figure 5. Simulation Models Landing Page

From there, the simulation model for this example can be found in the list, however there are many pages. A faster solution is to use the search box where the part number: FCD360N65S3R0 can be directly enter.

Document type:		
Product Taxonom	iy:	
Application:		

Figure 6. Simulation Models Search Box

Then, a list of models available appears.

Home > Support > Design Support > Design Resources & Do	cuments > Simulation Models	Previously	liewed Products
Simulation Models		Select Prod	uct ᅌ Go
Document type:			Clear Li
Product Taxonomy:		Technical	Support
- or - Enter Part#/Keyword Search		 > Email Teo > Live Tech > Phone Teo >> Find a Discovery 	chnical Support nical Support chnical Support stributor
Part #/keyword: FCD360N65S3R0 (Show Ali) 1 - 3 of 3 [1]			Page size: 50
Document Title 💌	Document ID/Size	Revision	Revision Date
LTspice model and symbol for SuperFET3 650V NEW	ONSEMI_SUPERFET3_650_Itspice.zip (282kB)	1.15	Mar, 2018
PSPICE model and symbol for SuperFET3 650V NEW	ONSEMI_SUPERFET3_650_pspice.zip (58kB)	1.15	Mar, 2018
SIMetrix SPICE model for SuperFET3 650V NEW	ONSEMI_SUPERFET3_650_simetrix.zip (71kB)	1.15	Mar, 2018

Figure 7. List of Available Models After Searching

Click on the link accordingly to the Spice simulator. It will start downloading the zip file containing all models available for the complete SuperFet 3 family.

Using the key word "SuperFET3", the list is much longer. Please, use the latest version available corresponding to the part used for this example.

As explain in James Victory's tutorial [1] at ON Semiconductor web site in the "Design Support > Technical Support > Tutorial" section, those models are developed for a particular technology with physical equations and not for one device by curve fitting.

From the Main Search Box

On the main page, type the part number partially (like FCD360N65 for example) and hit search.



Figure 8. Main Search Box

Then, the following results appears:

FCD360N65S3R0: N-Channel SuperFET®	III MOSFET, 650 V, 10 A, 360 mΩ		
V-Channel SuperFET® III MOSFET, 650 V, 10 A, ocuments: Application Notes , <u>Models</u> , Data She	, 360 mΩ N-Channel SuperFET® III MOSFET, 650 V eet , Package Drawing	, 10 A, 360 mΩ .	🛨
Sample/Buy Features Specification	one Package + Availability Beliability PCN	AEC/DDAD L	BoHS China BoH
lome > Products > Discrete > IGBTs & FETs > M ∃ hide	OSFETs > FCD360N65S3R0	A AEWFFAF II	NONO I CININA NON
ome > Products > Discrete > IGBTs & FETs > M ∃ hide Document Title	OSFETs > FCD360N65S3R0	Bevision	Revision Date
ome > Products > Discrete > IGBTs & FETs > Mi g hide Document Title Tspice model and symbol for SuperFET3 650V	Document ID/Size	Revision	Revision Date
tome > Products > Discrete > IGBTs & FETs > Mi	Document ID/Size ONSEMI_SUPERFET3_650_Itspice.zip (282kB) ONSEMI_SUPERFET3_650_itspice.zip (58kB)	Revision	Revision Date Mar, 2018 Mar, 2018

Figure 9. Main Search Box Result Page

Under the product name, click directly on "Models" and the models list table will appear.

Click on the link corresponding to the simulator platform. It will start downloading the zip file containing all models available for the complete SuperFet 3 family.

As explain in James Victory's tutorial [1] at ON Semiconductor web site in the "Design Support > Technical Support > Tutorial" section, those models are developed for a particular technology with physical equations and not for one device by curve fitting.

SIMULATION SETUP WITH SIMetrix

Preparing Files for SIMetrix

Unzip the file in a directory.

🗆] SIMetrix	-	Name *	Date modified	Туре	Size
ONSEMI_SUPERFET3_650_SIMETRIX		ONSEMI_SUPERFET3_650.sxslb	05/01/2018 17:12	SXSLB File	7 KE
Est		ONSEMI_SUPERFET3_650_simetrix_encrypted.txt	15/03/2018 15:53	Text Document	155 KE

Figure 10. Unzipping the Library Files

In this example, the library comes with two files:

- One symbols library file with a ".sxslb" extension.
- One sub-circuits' library file with a ".txt" extension. This file is encrypted.

WARNING: As the sub-circuits' library file is encrypted, it cannot be used with the free version of SIMetrix

SIMetrix will be ready to use those new models in all new schematics and simulations that will be create if the symbols' file and the sub-circuits file are included in SIMetrix environment as explained in the following paragraphs.

So, launch SIMetrix main program:

Dragging the sub-circuits' file to SIMetrix

In the file viewer, select the sub-circuits' text file and drag it to the command shell space.



Figure 11. Importing the Sub-Circuits' Text File

Then, SIMetrix asks what to do, select "Import" and click Ok. See below:



Figure 12. Importing Sub-Circuits Validation

Dragging the symbols' file to SIMetrix

In the file viewer, select the symbols' file and drag it to the command shell space.



Figure 13. Importing the Symbols' File

Then, SIMetrix asks what to do, select "OK".



Figure 14. Importing Symbols Validation

Dragging Both Files

In the file viewer, select the symbols file and the sub-circuits' text file and drag both files to the command shell space.

🖌 SIMetrix/SIMPLIS Main Wind	dow			
File View Help	D:\fg9vxb\Documents\FAE Training\SIMetrix\ONSEM	_SUPERFET3_650_SIM	ETRIX	
	GOV JANNETIX + ONSEMI_SUPERFET3_650_SIME	TRIX	▼ 🐼 Search ONS	
File View	File Edit View Tools Help		,	
Add Directory	Organize 🔻 📄 Open Share with 👻 E-mail Burn	New folder		
Seminar SIMetrix	Name ^		Date modified	Туре
	ONSEMI_SUPERFET3_650.sxslb		05/01/2018 17:12	SXSLB File
		_encrypted.txt	15/03/2018 15:53	Text Document
	🚺 SIMe			
	10 🔟			
				Þ
	2 items selected Date modified: 05/01/2018 17	12 - 15/0 Date created:	05/01/2018 17:12 - 15,	/03/2018 15:53
	Size: 160 KB			
Command Shell	2 items selected		j Comp	uter //.
<u></u>				
	Cre	ate New		
Welcome to SIMetrix/SIME Version 8.00r (x64)	PLIS Pro	Schematic		
		Symbol		

Figure 15. Importing the Sub-Circuits' Text File and the Symbols File Together

The same windows as Figures 12 and 13 will appear to confirm these actions. Click Ok for both windows.

Adding the sub-circuits' file using SIMetrix menu

To include the sub-circuits for the simulation: from the menu, got to "File -> Model Library -> Add/Remove libraries..." to select the directory where the sub-circuits file is located and click "Select Folder". See below:

🖌 Choose Directory			×
⊙ ⊽ <u></u> •	✓ My Documents	👻 🌆 Search SIMetr	ix 🛛 🙋
Organize 👻 New fold	er		= • 🔞
🍌 SIMetri	ix Anne ^	Date modified	Туре
) ONSE		03/05/2018 13:41	File folder
	-		
	<u> </u>		Þ
F	older: ONSEMI_SUPERFET3_650_SIMETRIX		
		Select Folder	Cancel

Figure 16. Selecting Sub-Circuits' Text File Directory

The following sub-circuits' library manager window opens with the library waiting to be added:



Figure 17. Sub-Circuits Library Waiting to be Included

Then, select the available library and select the up arrow as shown:

C: \Program File C: \Program File	s\SIMetrix800 s\SIMetrix800	\support\Mode \support\Mode	els*.lb els\digital*.ll	Ь		4
						-
		+ + <		Select	t	
vailable Libraries	·					
		SIMetri	X\ONSEMI_S	UPERFET3_6	50_SIMETRI	.X*.b

Figure 18. Library Selection and Inclusion

At the end, the sub-circuits' library manager should look like this:

Currently Selected Libraries	0)aupport/Meda	a* a		 • 1
C:\Program Files\SIMetrix80	0\support\Mode 0\support\Mode 3\SIMetrix	Is\~.Ib Is\digital*.Ib (\ONSEMI_SUF	PERFET3_650_S	₽ ↓
Available Libraries	† †			
				-

Figure 19. Sub-Circuits' Library Text File Added

Adding the Symbols' File Using SIMetrix Menu

To include the symbols for the schematic: from the menu, go to "File -> Symbol Manager..." to open the symbol manager as shown below:

analog, sxslb	A 🔺	Symbols	
analog_devices_all.sxslb			
digital.sxslb			
psu.sxslb			
connection.sxslb	Add		
maxim_controllers.sxslb	Auu		
miscellaneous.sxslb			
mixed_mode.sxslb	Remove		
passives.sxslb			
semiconductors.sxslb	Create		
sheet_text.sxslb			
sources.sxslb			
simplis.sxsib			
WebSim_Connect.sxsib	-		
Intersii controllers.sxsib			
· · · + · · · · + · · · · + · · · · + ·	*····*		
		Delete	Conv Create,
		Delete	Copy, Create

Figure 20. Symbol Manger to Include the Symbols' Library

Look for the file with the ".sxslb" extension, select the file and click "Open".



Figure 21. Selecting Symbols' File

The symbols are now added to the library as shown below. The symbol manager can be closed.



Figure 22. Symbols Added in the Symbols' Library

SIMULATION SETUP WITH OrCAD PSpice

There is two ways to setup the library in OrCAD:

- 1. Local setting in the PSpice project.
- 2. Global setting for all PSpice projects.

The local setting needs to be done for each simulation project while creating the simulation project. The global setting needs to open a blank or dummy project to access the menu items.

Adding file to OrCAD PSpice library directory

Unzip the file in a directory.

🅌 Test	-	Name *	Date modified	Туре	Size	
🔒 LIB		ONSEMI_SUPERFET3_650.OLB	15/03/2018 16:26	OLB File	155 KB	
ONSEMI_SUPERFET3_650_PSPICE		ONSEMI_SUPERFET3_650.bxt	15/03/2018 15:53	Text Document	315 KB	

Figure 23. Unzipping the Library Files

In this example, the library comes with two files:

- One symbols library file with a ".OLB" extension
- One sub-circuit library file with a ".txt" extension. This file is encrypted.

The symbols file could be added to the capture tool library directory and the sub-circuits library file to the PSpice library directory. In those two locations, create an ON directory in each one to store the files.

Depending on the version, the symbol library directory is something like: "C:\Cadence\....\tools\capture\library\pspice". In this location, create an ON directory. Then, copy the symbol file with a ".OLB" extension to in the ON directory. It should look like this:

🖃 💒 Local Disk (C:)		Name *		Date mo	dified	Туре	Size	
□) Cadence □) SPB_17.2) bin		ONSEMI_SUPERFI	ET3_650.OLB	15/03/2	018 16:26	OLB File		155 KB
🕀 🔒 doc								
🕀 🕌 share								
🖃 퉲 tools								
🕀 퉲 bin								
🖃 퉲 capture								
🖃 🍌 library								
🖃 🍌 pspice								
🍌 advanis								
🕀 🏬 modeled								
📙 ON								

Figure 24. Copying ON Symbol Library File in OrCAD Capture Library Directory for Pspice

Depending on the version, the symbol library directory is something like: "C:\Cadence\....\tools\pspice\library". In this location, create an ON directory. Then, copy the sub-circuit file with a ".txt" extension to in the ON directory. It should look like this:

🚢 Local Disk (C:)		Name ^	Date modified	Туре	Size
🔒 Cadence			15/03/2018 15:53	Text Document	215 KB
J SPB_17.2			15/05/2018 15:55	Text Document	313 KD
in 🔒 bin					
July doc					
🗼 share					
lools					
鷆 bin					
🍌 capture					
📙 cdnshelp					
🌗 fet					
🔒 InstallUtils					
ib lib					
🌗 msbase					
🔰 pspice					
🍑 api					
Capture_samples					
Common					
)]] library					
🍌 discretetables					
) distribution					
Jk ON	-				

Figure 25. Copying ON Sub-Circuit Library File in OrCAD PSpice Library Directory



Setting the library as local for one project

Creating a blank PSpice project with the symbols for the schematic capture

From the main menu, choose: "File -> New -> Project..." to create a new project file (with one blank schematic page) in the working directory for example. The following window opens:



Figure 26. New Project Window Setup

Give a name to the project and a location, select "PSpice Analog or Mixed A/D" type, and then, hit OK.

A new window appears and select "Create a blank project" and hit OK.

Create PSpice Project		
C Create based upon an existing project		OK
AnalogGNDSymbol.opj	*	Browse
G. Crasta a blank assist		Cancel
 Create a brank project 		Help

Figure 27. New Project Setup

Select the project tree tab (instead of the schematic first page). Then, expand the "PSpice Resources" branch. See below:



Figure 28. Complete Project Tree

Right click on the "Library" branch and select "Add file". Locate the ".OLB" file from the ON Semiconductor symbol library directory created.

In this example, the file is: "ONSEMI_SUPERFET3_650.OLB" in the "C:\Cadence\....\tools\capture\library\pspice\ON" directory. The result should look like:



Figure 29. ON Symbol Library Loaded in the Project

Creating a blank Simulation Profile with the sub-circuits for simulation

From the main menu, choose: "PSpice -> New Simulation Profile" to create a new simulation setup that will include all sub-circuits' definitions that all future simulations will inherit. The following window opens:

ew Simulation	
Name:	Custa
ON_Library_Included	Liedle
Inherit From:	Cancel
none	

Figure 30. New Simulation Profile to Include Sub-Circuits' Definitions

On the "Analysis" tab, select "Bias Point", however this is not mandatory.

	Analysis Type: Bias Point Deptions: General Settings Temperature (Sweep) Save Bias Point Load Bias Point	Output File Options Data Concettoring Processing Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP) Perform Sensitivity analysis (.SENS) Output variable(s): Calculate small-signal DC gain (.TF) From Input source name: To Output variable:
--	--	--

Figure 31. Bias Point Simulation as Default Profile

Go to the "Configuration Files" tab, then select "Library" as a file category to include.

General	Analysis	Configuration Files	Options	Data Collection	Probe Window	
Category:		Filename:				
Stimulus	~					Browse
Library	(1)	Configured Files		ſ	× + +	
nciude				0		
-	ndov					Add as Global
	Index					Add to Design
						Add to Profile
						Edit
						Change
		Library Path				
		"C:\Cadence\SPE	_17.2\tools\	PSpice\Library"		Browse

Figure 32. Simulation Setting Configuration Files Tab

Locate the text file that includes all sub-circuits' definitions by clicking on "Browse".

In this example, the file is: "ONSEMI_SUPERFET3_650.txt" in the "C:\Cadence\....\tools\pspice\library\ON" directory. Pay attention to select "All files" type of file in order to locate the text file. By default, it locates only ".lib" files.

👫 Open					×
Look in	n: 🚺 ON		- 🗿 🗊 🛙	• 🖽 🍤	
3	Name A	UPERFET3_650.txt	✓ Date modified 15/03/2018 15:53	 ▼ Type 3 Text Docum 	− ent
Recent Places					
Desktop					
Libraries					
Computer					
Network					
	•				•
	File name:	ONSEMI_SUPERF	ET3_650.txt	• 0	pen
	Files of type:	All Files (*.*)		- <u>Ca</u>	incel

Figure 33. Selecting the Text File Containing the Sub-Circuits' Definitions

Then, click "Open" to add the file to the simulation profile. The result should look like:

General	Analysis	Configuration Files	Options	Data Collection	Probe Window		
Category:		Filename:					
Stimulus		C:\Cadence\SPE	Browse				
Library Include		Configured Files					
Update Index		C:\Cadence ERFET3_65	Add as Global				
		Add to Design					
			Add to Profile				
		Edit					
						Change	
		Library Path					
		"C:\Cadence\SPB_17.2\tools\PSpice\Library" Browse					

Figure 34. New Simulation Profile Setup With Sub-Circuits' Definitions Included

Click "Add to Design" to add the file in the middle window list space. Click "Apply" and Ok before closing the window.

At the end, the project tree should look like this:



Figure 35. Project Tree With All ON Library Files

Setting the library files as global for all projects

From the main menu, choose: "File -> New -> Project..." to create a new dummy project file (with one blank schematic page) in the working directory for example. The following window opens:



Figure 36. New Project Window Setup

Give a name to the project and a location, select "PSpice Analog or Mixed A/D" type, and then, hit OK. A new window appears and select "Create a blank project" and hit OK.

Create PSpice Project		>
C Create based upon an existing project		ОК
AnalogGNDSymbol.opj	v.	Browse
		Cancel
 Create a blank project 		Help

Figure 37. New Project Setup

This project is not a real schematic. This is just to have access to the schematic commands and be able to setup the global assignment needed to setup the library that will be used in all future projects. This project can be delete or cancel (or not save) at the end.

Setting symbols for the schematic capture environment

From the main menu, choose: "Place -> Part..." to open the library browser panel. In that panel, locate the "Add Library" icon/button and click it. See below:

Place	e Part	ţ.	- X
Pa	t 🔳 🥥		
Par	t List: 🍸		
	Add Library		
Lib	raries:	1	
D	esign Cache		

Figure 38. Add Library Button

OrCAD will ask to locate the symbol library file, locate it in the ON directory created and click Open. This will add the symbol to the capture environment. See the result below:

Part)	
				-	
Part List:			Y		
FCB070N65S3_3p FCB070N65S3_5p FCB199N65S3_5p FCB199N65S3_5p FCB260N65S3_5p FCB260N65S3_5p FCD260N65S3_5p FCD260N65S3_5p					
Libraries:	C	<u>ا المج</u>	×		
Design Cache	3 650				
Solution Solution					

Figure 39. Symbol Library File Added as Global

Setting sub-circuits for PSpice simulation environment

From the main menu, choose: "PSpice -> New Simulation Profile" to create a new dummy simulation setup. A name must be given to that simulation profile. See Figure 30.

Go to the "Configuration Files" tab, then select "Library" as a file category to be included. Locate the text file that includes all sub-circuits' definitions by clicking on "Browse". See Figure 32.

In this example, the file is: "ONSEMI_SUPERFET3_650.txt" in the "C:\Cadence\....\tools\pspice\library\ON" directory. Pay attention to select "All files" type of file in order to locate the text file. By default, it locates only ".lib" files. See Figure 33.

Then, click "Open" to add the file to the simulation profile.

The result should look like:

General	Analysis	Configuration Files	Options	Data Collection	Probe Window		
Category:		Filename:					
Stimulus		C:\Cadence\SPE	C:\Cadence\SPB_17.2\tools\pspice\library\ON\ONSEMI_SUPERFI				
Library Include		Configured Files		(× + +		
💟 Update I	ndex	ERFET3_6	50.txt			Add to Design Add to Profile Edit Change	
		Library Path					
		"C:\Cadence\SPE	3 17.2\tools\	PSpice\Library"		Browse	

Figure 40. New Simulation Profile Setup With Sub-Circuits' Definitions Included

Click "Add as Global" to add the file to list space in the window middle. Click "Apply" and Ok before closing the window. Then, close without saving the project.

SIMULATION SETUP WITH LTSpace

Adding Files to LTSpice Library Directory

Unzip the file in a directory.

🖃 🅌 LTSpice	1	Name *	Date modified	Туре	Size
ONSEMI_SUPERFET3_650_LTSPICE Symbol		📙 Symbol	15/03/2018 15:56	File folder	
Jan Symbol		ONSEMI_SUPERFET3_650_ltspice.txt	15/03/2018 15:56	Text Document	594 KB

Figure 41. Unzipping the Library Files

In this example, the library comes with one file and one directory:

- One symbols' library directory named "Symbol"
- One sub-circuits' library file with a ".txt" extension. This file is encrypted.

In the symbols' directory, there are two files per part number: one with and one without thermal network.

E 📕 LTSpice	▲ Name *	Date modified	Туре	Size
Worker SuperFet3_650_LTSPICE	desktop.ini	04/05/2017 14:56	Configuration settings	1 K
Symbol	D- FCB070N65S3_3p.asy	16/03/2016 12:30	LTspice Symbol	18
	FCB070N65S3_5p.asy	16/03/2016 12:30	LTspice Symbol	1 K
	D- FCB199N65S3_3p.asy	03/05/2017 10:44	LTspice Symbol	11
	D- FCB199N65S3_5p.asy	03/05/2017 10:44	LTspice Symbol	18
	D-FCB260N65S3_3p.asy	06/09/2017 11:49	LTspice Symbol	1 8
	D- FCB260N65S3_5p.asy	06/09/2017 11:49	LTspice Symbol	11
	D- FCD260N65S3_3p.asy	08/08/2017 14:02	LTspice Symbol	11
	D- FCD260N65S3_5p.asy	08/08/2017 14:02	LTspice Symbol	18
	D- FCD360N65S3R0_3p.asy	29/11/2017 11:35	LTspice Symbol	18
	D-FCD360N65S3R0_5p.asy	29/11/2017 11:35	LTspice Symbol	11

Figure 42. All Symbols Available

The symbol with thermal network has two more pins than the one without.

Locate the location of the library files are by creating a blank schematic and trying to place a part. The following window appears where the path can be seen.

\Documents\L	TspiceXVII\lib\sym	<u> </u>
_	Library	s test fixture
bv cap csw current diode e e f FernteBead FernteBead2 g	sym\ g2 h ind iN016750-2 ISO16750-2 ISO7637-2 LED load load2 load2 lpnp tline	mesfet nif nmos npn3 npn3 npn4 pif pmos pmos4
	nenta L Tapice XVII Nib bv cap csw current diode e e f FentleBead FentleBead S FentleBead 2 g	Penta L Tspice XVII Vib \sym\ bv g2 cap h csw ind current ind2 diode ISO 16750-2 e ISO 7637-2 e2 LED f load Fernte Bead load2 Fernte Bead2 long g thine

Figure 43. Library Path

In this example, the library is in the path "D:\....\Documents\LTspiceXVII\lib". The "sym" directory is the one where the symbols' files are stored. There are 3 sub-directories in the "lib" directory. The other important one is the "sub" directory that stores all sub-circuits' files that define sub-circuits statement used in simulations' schematics.

Create an ON directory in the "D:\....\lib\sub" sub-circuits' directory and in the "D:\....\lib\sym" symbols' directory to store respectively the sub-circuit ".txt" file and the ".asy" symbols' files. Then, copy the ".txt" file to the "D:\....\lib\sub\ON" directory and all ".asy" files to "D:\....\lib\sym\ON" directory.



Figure 44. ON Sub-Directories for Sub-Circuit and Symbols' Files

Special access to LTSpice library folder on the MAC version

The LTSpice MAC version has its library in the hidden user library folder. To access that folder, use the finder menu. Click and maintain the mouse button down on the "Go" menu and press "alt" or "¬" (for alternate) and a new menu item in the "Go" menu list will appear. It is the "Library" item. Go to that item to open the user "Library" folder. Inside that folder go down to "Application Support" folder and down to "LTspice" folder. The LTSpice "Library" folder appears with the "sub", "sym" and "cmp" sub–folders.



Figure 45. Library Folder Access on a MAC

As the "lib" folder and subfolders are inside the user library folder, LTSpice MAC cannot write directly in it. Files can be copied in it with administrator access but not edited with LTSpice inside those folders. So, when editing library file with LTSpice (as proposed later), files should first be saved them in the user space and then copy them to the library space.

However, a text editor can write in this library space.

Setting the library in LTSpice

There is two ways to setup the library in LTSpice:

- 1. Local setting in the simulation file.
- 2. Global setting in the symbol files.

The local setting is very fast and easy but need to be done for each simulation schematic.

The global is a little bit cumbersome because it needs to edit each symbol files that will be used. If a new version of the library shows up, this needs to be redone by editing each symbol file or rename the sub-circuit file to have the same name. The revision number in the file name is lost but it is included in the sub-circuit file.

Creating a blank schematic with the library statement

Launch LTSpice: *J* and create a new schematic.

In the schematic, add a spice directive with the statement: ".Lib ON\LibraryFileName.txt". It assumes all sub-circuits' files are located in a directory starting at "D:\....\lib\sub" in that example. So, only the relative path to the library need to be added.



Figure 46. Library Call Simulation Statement

This very easy but this ".Lib" statement needs to be added to each new schematic.

Editing symbols to link them to the library

Locate the symbol file for the part used for this example: the FCD360N65S3R0 SupetFET 3 MOSFET without the thermal network. So, locate the file: "FCD360N65S3R0_3p.asy". Double–click on that file to launch LTSpice and edit the symbol. A specific menu items is available for the symbol editor but it is the same toolbar buttons.



Figure 47. LTSpice Symbol Editor

Go to "Edit -> Attributes -> Edit Attributes" or hit "Ctrl-A". The attribute window opens. There is a specific attribute named: "ModelFile" that can store the link to the sub-circuits' definitions' file. Just add the relative path. So, it should be something similar to "ON\LibraryFileName.txt". See blow:

	1
attribute	value
renx Nata Madal	^
piceiviodei	ECD300N(EEC3D0 2-
/alue	PCD360N6353R0_3p
aluez	
pice Line 2	
Description	N Channel MOSEET transister
AndelCle	ONLONGEMU SUBEREET2 CE0 Arrise ht
lodelfile	UN/UNSEMI_SUPERFEI3_650_ttspice.bt

Figure 48. Add the Sub-Circuits' Definitions' File Path to the Symbol

Then, click OK, save the symbol. Quit and re–launch LTSpice: *I* to refresh LTSpice library settings. Otherwise, the modifications will not be taken into account.

When the symbol has been modified, to add the ".Lib" simulation statement in simulation schematic files as in the previous chapter: "Creating a blank schematic with the library statement." is not anymore needed. This method is more complex and need more time because it needs to be done for each symbol, but when done with all symbols, no need to take care how to use the part. It comes with the link to the sub-circuit definition.

Fast symbols' files editing with a text editor

As symbol files are text files, it is possible to edit them directly with any raw text editor.

So, a faster way is to copy and paste the following statement that indicate the sub-circuit definition file location:

"SYMATTR ModelFile ON\ONSEMI_SUPERFET3_650_ltspice.txt"

For the MAC version, add the following statement anywhere in the text symbol file link symbol and sub-circuit definition: "SYMATTR ModelFile ~/Library/Application Support/LTspice/lib/sub/ON/ONSEMI_SUPERFET3_650_ltspice.txt"

OR :

"SYMATTR ModelFile ON/ONSEMI SUPERFET3 650 ltspice.txt"

When finished, the FCD360N65S3R0 symbol file should look like this one with the ModelFile attribute at the end of the file:

ī.

Version 4	WINDOW 3 56 72 Left 2
SymbolType CELL	SYMATTR Value FCD360N65S3R0_3p
LINE Normal 48 48 48 96	SYMATTR Prefix X
LINE Normal 16 80 48 80	SYMATTR Description N-Channel MOSFET
LINE Normal 40 48 48 48	transistor
LINE Normal 16 48 40 44	PIN 48 0 NONE 0
LINE Normal 16 48 40 52	PINATTR PinName D
LINE Normal 40 44 40 52	PINATTR SpiceOrder 1
LINE Normal 16 8 16 24	PIN 0 80 NONE 0
LINE Normal 16 40 16 56	PINATTR PinName G
LINE Normal 16 72 16 88	PINATTR SpiceOrder 2
LINE Normal 0 80 8 80	PIN 48 96 NONE 0
LINE Normal 8 16 8 80	PINATTR PinName S
LINE Normal 48 16 16 16	PINATTR SpiceOrder 3
LINE Normal 48 0 48 16	SYMATTR ModelFile ON\ONSEMI_SUPER-
WINDOW 0 56 32 Left 2	FET3_650_ltspice.txt

Figure 49. LTSpice Symbol Text File Example With the ModelFile Statement

SIMULATE WITH ON PHYSICAL AND SCALABLE COMPONENT MODELS

In this example, simulation will be used to measure the on resistance at 1 A drain current and 10 V gate voltage. More complex simulations can be run but this is not the purpose here.

Schematic capture

With SIMetrix

Select the FCD360N65S3R0 SuperFET3 device and make the following simple schematic. Use the 3 pins version that doesn't include the thermal network.



Figure 50. ON Resistance Measurement Setup With SIMetrix

By reading the drain voltage with a voltage probe, the resistance in Volts will be directly shown because 1 A is applied.

With OrCAD

Select the FCD360N65S3R0 SuperFET3 device and make the following simple schematic. Use the 3 pins version that doesn't include the thermal network.



Figure 51. ON Resistance Measurement Setup With OrCAD Pspice

By reading the drain voltage with a voltage probe, the resistance in Volts will be directly shown because 1A is applied.

With LTSpice

Select the FCD360N65S3R0 SuperFET3 device and make the following simple schematic. Use the 3 pins version that doesn't include the thermal network.



Figure 52. ON Resistance Measurement Setup With LTSpice

By reading the drain voltage with a voltage probe, the resistance in Volts will be directly shown because 1 A is applied.

Simulation setup

With SIMetrix

Create a basic time domain simulation profile for 10 us. The default settings can be used here. Click "Apply" and Ok.

Choose Analysis				×
Transient AC DC	C Noise	TF SOA Data	Options	Analysis Mode
Stop time		100		
Data output options -				
Start data output @	0		Default	
PRINT step Output all data	200n	C Output at .PRIN	T step	

Figure 53. Simulation Analysis Selection With SIMetrix

With OrCAD

Create a new simulation profile that is inherited from the one created before if local library setting is used or a new simulation profile without heritage if global library setting is used. This will automatically include all sub-circuits' definitions. If several simulation profiles can be run on the same schematic, no need to add the files each time a new profile is created.

New Simulation	×
Name:	Crasta
Measure-RdsON	create
Inherit From:	Cancel
SCHEMATIC1-ON_Library_Includ	
Root Schematic: SCHEMATIC1	

Figure 54. Inherit Simulation Profile Creation With OrCAD Pspice

Create a basic time domain simulation profile. The default settings can be used here. Click "Apply" and Ok.

With LTSpice

Create a new simulation profile from the menu: "Simulate \rightarrow Edit Simulation Cmd". Create a basic "Transient" simulation for 10 μ s for example.

	AL Analysis	DC sweep	Noise	DC Transfer	DC op pnt
	Perform a non-linear, time-domain simulation.				
	Stop time:			10us	
	Time to start saving data:				
	Maximum Timestep:				
	Start external DC supply voltages at 0V:				
	Stop simulating if steady state is detected:				
	Don't reset T=0 when steady state is detected:				
	Step the load current source:				
	9	kip initial ope	erating po	int solution: I	

Figure 55. New Simulation Command Creation With LTSpice

Click Ok and place the statement in the schematic.

Results

With SIMetrix

Then, just run this new time domain simulation.



Figure 56. Simulation Results Obtained With SIMetrix

Using the cursor to read the R_{dsON} value that should be 298.9 m Ω with 1 A drain current and 10 V gate voltage.

With OrCAD

Then, just run this new time domain simulation.



Figure 57. Simulation Results Obtained With OrCAD Pspice

Read the R_{dsON} value that should be 298.9 m Ω with 1 A drain current and 10 V gate voltage.

With LTSpice

Then, just run this new time domain simulation.



Figure 58. Simulation Results Obtained With LTSpice

Place a marker on the curve to measure a 298.9 m Ω R_{dsON} with 1 A drain current and 10 V gate voltage.

CONCLUSION

This paper shows how easy it is to use ON Semiconductor physical and scalable components' models available in ON Semiconductor web site for SIMetrix, OrCAD Pspice or LTSpice.

Generally, simulation is used to predict, understand, and analyze a system. The models are primarily made for this purpose. Due to model internal structure explained in [1], the physical and scalable models can be used to exract device parameters that are not given in the data sheet and corresponding to the system operating point.

BIBLIOGRAPHY

- "Physically Based, Scalable SPICE Modeling Methodologies for Modern Power Electronic Devices" or "SPICE Modeling Tutorial", James Victory, ON Semiconductor European Power Seminar 2017, http://www.onsemi.com/pub/Collateral/TND6248–D.PDF
- 2. https://www.simetrix.co.uk/index.html
- 3. <u>http://www.orcad.com</u>
- 4. http://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

All brand names and product names appearing in this document are registered trademarks or trademarks of their respective holders.

ON Semiconductor and are trademarks of Semiconductor Components Industries, LLC dba ON Semiconductor or its subsidiaries in the United States and/or other countries. ON Semiconductor owns the rights to a number of patents, trademarks, copyrights, trade secrets, and other intellectual property. A listing of ON Semiconductor's product/patent coverage may be accessed at <u>www.onsemi.com/site/pdf/Patent-Marking.pdf</u>. ON Semiconductor reserves the right to make changes without further notice to any products herein. ON Semiconductor makes no warranty, representation or guarantee regarding the suitability of its products for any particular purpose, nor does ON Semiconductor assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation special, consequential or incidental damages. Buyer is responsible for its products and applications using ON Semiconductor products, including compliance with all laws, regulations and safety requirements or standards, regardless of any support or applications information provided by ON Semiconductor. "Typical" parameters which may be provided in ON Semiconductor data sheets and/or specifications can and do vary in different applications and actual performance may vary over time. All operating parameters, including "Typicals" must be validated for each customer application by customer's technical experts. ON Semiconductor does not convey any license under its patent rights or the rights of others. ON Semiconductor and tesigned, intended, or authorized for use as a critical component in life support systems or any FDA Class 3 medical devices or medical devices with a same or similar classification in a foreign jurisdiction or any devices intended for implantation in the human body. Should Buyer purchase or use ON Semiconducts harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or deat

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:

Literature Distribution Center for ON Semiconductor 19521 E. 32nd Pkwy, Aurora, Colorado 80011 USA Phone: 303-675-2175 or 800-344-3860 Toll Free USA/Canada Fax: 303-675-2176 or 800-344-3867 Toll Free USA/Canada Email: orderlit@onsemi.com N. American Technical Support: 800–282–9855 Toll Free USA/Canada Europe, Middle East and Africa Technical Support: Phone: 421 33 790 2910 ON Semiconductor Website: www.onsemi.com

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative