AND9783/D

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

ON Semiconductor®

www.onsemi.com

APPLICATION NOTE

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: http://www.onsemi.com/PowerSolutions/product.do?id=FCD360N65S3R0

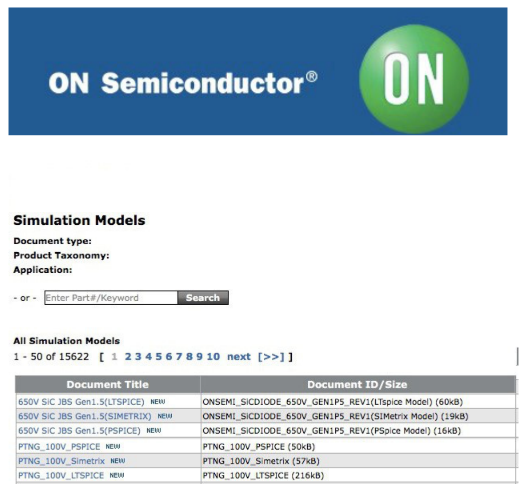


Table of Contents

Introduction . . . 1
Obtaining the models . . . 4
From the product page . . . 4
- From the model landing page . . . 5
- From the main search box . . . 7

Simulation setup with SIMetrix . . . 8
- Preparing files for SIMetrix . . . 8
- Dragging the sub−circuits' file to SIMetrix . . . 8
- Dragging the symbols' file to SIMetrix . . . 9
- Dragging both files . . . 10
- Adding the sub−circuits' file using SIMetrix menu . . . 10
- Adding the symbols' file using SIMetrix menu . . . 12

Simulation setup with Orcad PSpice . . . 14
- Adding file to OrCAD PSpice library directory . . . 15
- Setting the library as local for one project . . . 15
  - *Creating a blank PSpice project with the symbols for the schematic capture* . . . 15
  - *Creating a blank Simulation Profile with the sub−circuits for simulation* . . . 16
- Setting the library files as global for all projects . . . 19
  - *Setting symbols for the schematic capture environment* . . . 20
  - *Setting sub−circuits for PSpice simulation environment* . . . 21

Simulation setup with LTspice . . . 22
- Adding files to LTSpice library directory . . . 22
- Special access to LTSpice library folder on the MAC version . . . 23
- Setting the library in LTSpice . . . 24
  - *Creating a blank schematic with the library statement* . . . 24
  - *Editing symbols to link them to the library* . . . 24
  - *Fast symbols' files editing with a text editor* . . . 26

Simulate with ON physical and scalable component models . . . 27
- Schematic capture . . . 27
  - *With SIMetrix* . . . 27
  - *With OrCAD* . . . 27
  - *With LTSpice* . . . 28
- Simulation setup . . . 28
  - *With SIMetrix* . . . 28
  - *With OrCAD* . . . 28
  - *With LTSpice* . . . *29*
- Results . . . 29
  - *With SIMetrix* . . . 29
  - *With OrCAD* . . . 30
  - *With LTSpice* . . . 30

Conclusion . . . 31
Biography . . . 31

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 10
Figure 16. Selecting sub−circuits' text file added 10
Figure 17. Sub−circuits' library waiting to be included 11
Figure 18. Library selection and inclusion 11
Figure 19. Sub−circuits' library text file added 11
Figure 20. Symbol Manager to include the symbols' library 12
Figure 21. Selecting symbols' file 13
Figure 22. Symbols added in the symbols library 13
Figure 23. Unzziping the library files 14
Figure 24. Copying ON symbols library file in OrCAD Capture library directory for PSpice 14
Figure 25. Copying ON sub−circuit library file in OrCAD Pspice library directory 15
Figure 26. New project window setup 15
Figure 27. New project setup 16
Figure 28. Complete project tree 16
Figure 29. ON symbol library loaded in the project 16
Figure 30. New simulation profile to include sub−circuits' definitons 16
Figure 31. Bias point simulation as default profile 17
Figure 32. Simulation setting configuration files tab 17
Figure 33. Selecting the text file containing the sub−circuits' definitions 18
Figure 34. New simulation profile setup with sub−circuits' definitions included 18
Figure 35. Project tree with all ON library 19
Figure 36. New project window setup 19
Figure 37. New project setup 19
Figure 38. Add library button 20
Figure 39. Symbol library file added as global 20
Figure 40. New simulation profile setup with sub−circuits' definitons included 21
Figure 41. Unzipping the library files 22
Figure 42. All symbols available 22
Figure 43. Library path 22
Figure 44. ON sub−directories for sub−circuits' definitons included 23
Figure 45. Library folder access on a MAC 23
Figure 46. Library call simulation statement 24
Figure 47. LTSpice symbol editor 24
Figure 48. Add the sub−circuits' definitions' file path to the symbol 25
Figure 49. LTSpice symbol text file example with the ModelFile statement 26
Figure 50. ON resistance measurement setup with SIMetrix 27
Figure 51. ON resistance measurement setup with OrCAD PSpice 27
Figure 52. ON resistance measurement setup with LTSpice 28
Figure 53. Simulation analysis selection with SIMetrix 28
Figure 54. Inherit simulation profile creation with OrCAD PSpice 28
Figure 55. New simulation command creation with LTSpice 29
Figure 56. Simulation results obtained with SIMetrix 29
Figure 57. Simulation results obtained with OrCAD PSpice 30
Figure 58. Simulation results obtained with LTSpice 30

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:

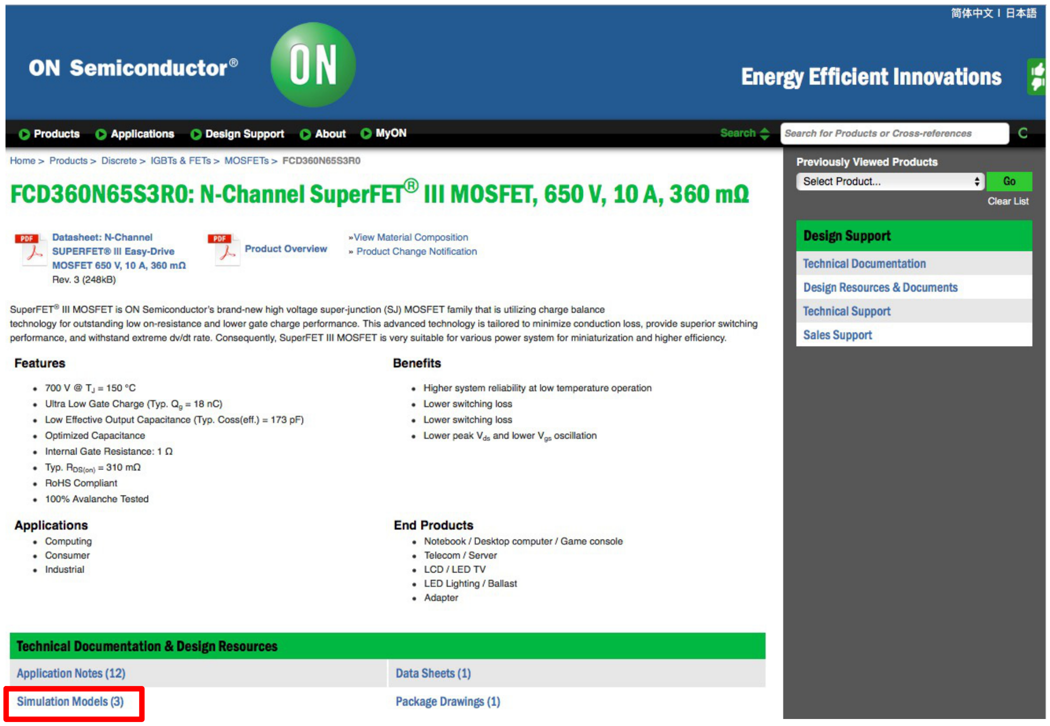


Figure 1. Online Product Page

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.

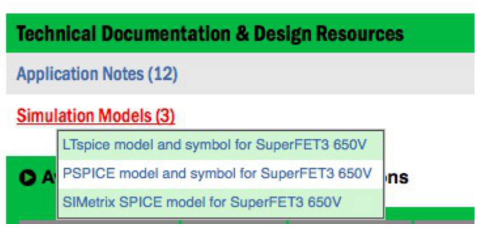


Figure 2. Links to Models Available from Product Page

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:

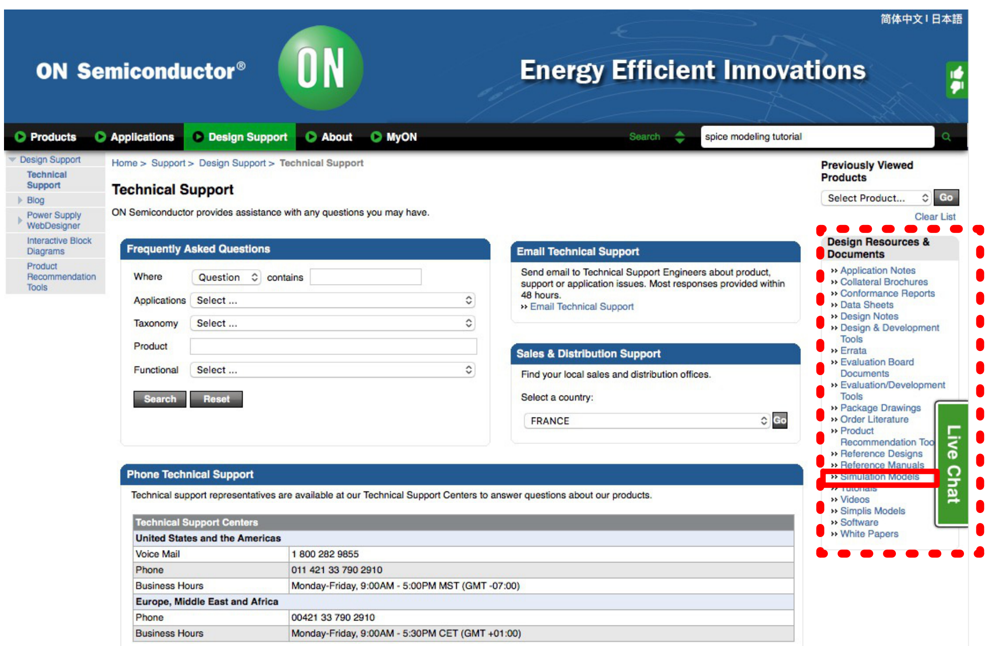


Figure 4. Technical Support Landing Page

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:

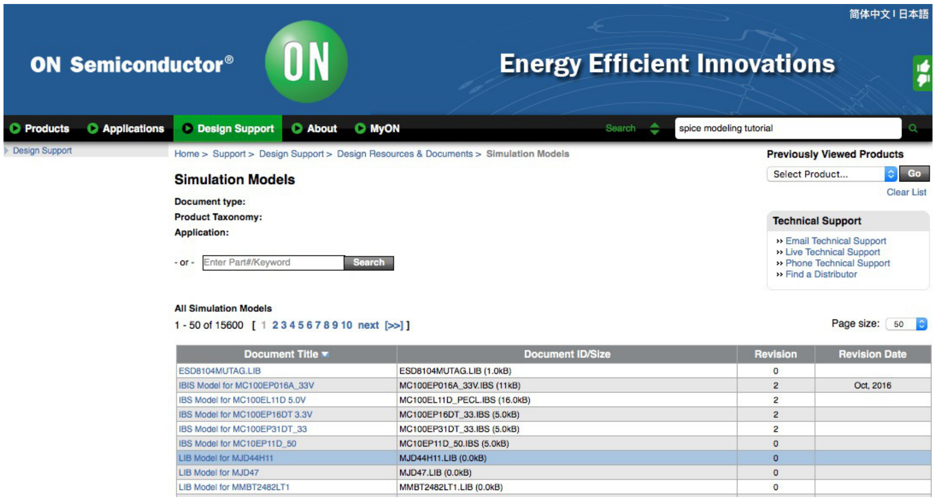


Figure 5. Simulation Models Landing Page

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.

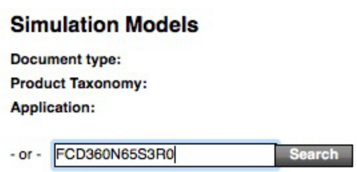


Figure 6. Simulation Models Search Box

Then, a list of models available appears.

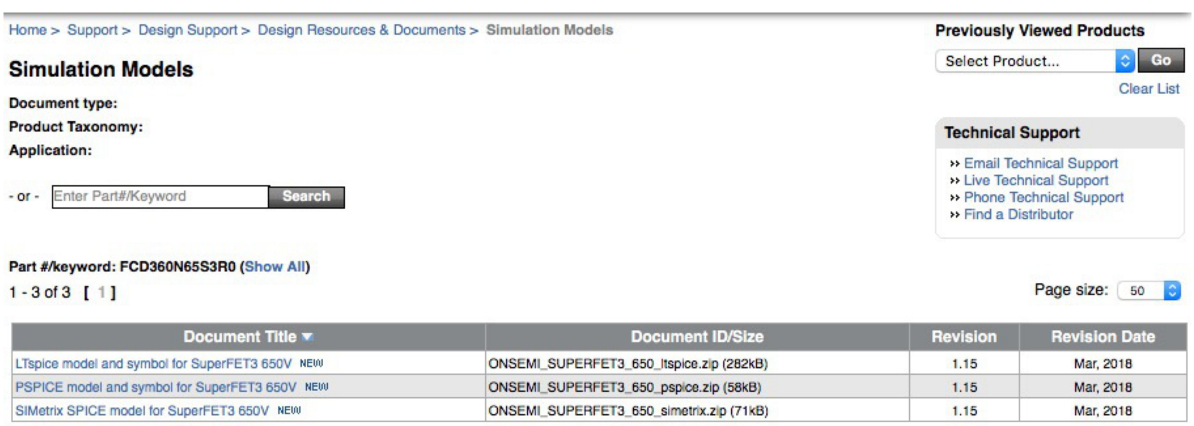


Figure 7. List of Available Models After Searching

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:

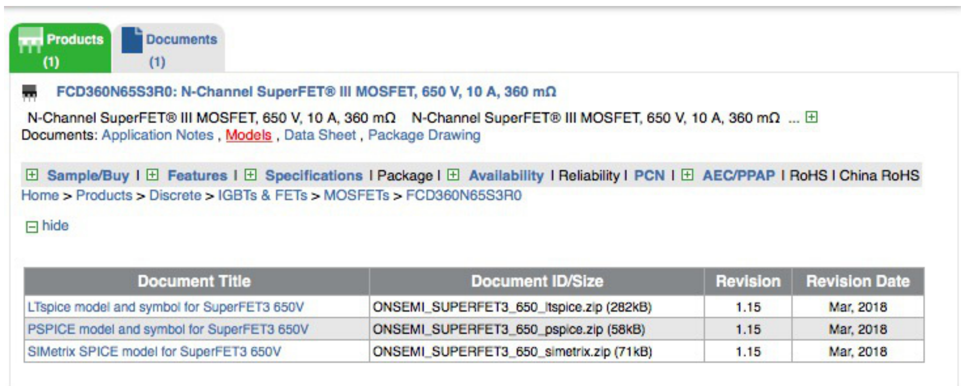


Figure 9. Main Search Box Result Page

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.

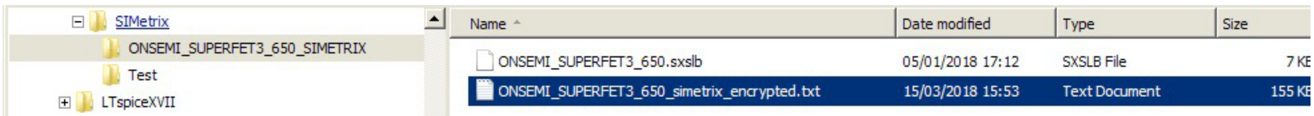


Figure 10. Unzipping the Library Files

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.

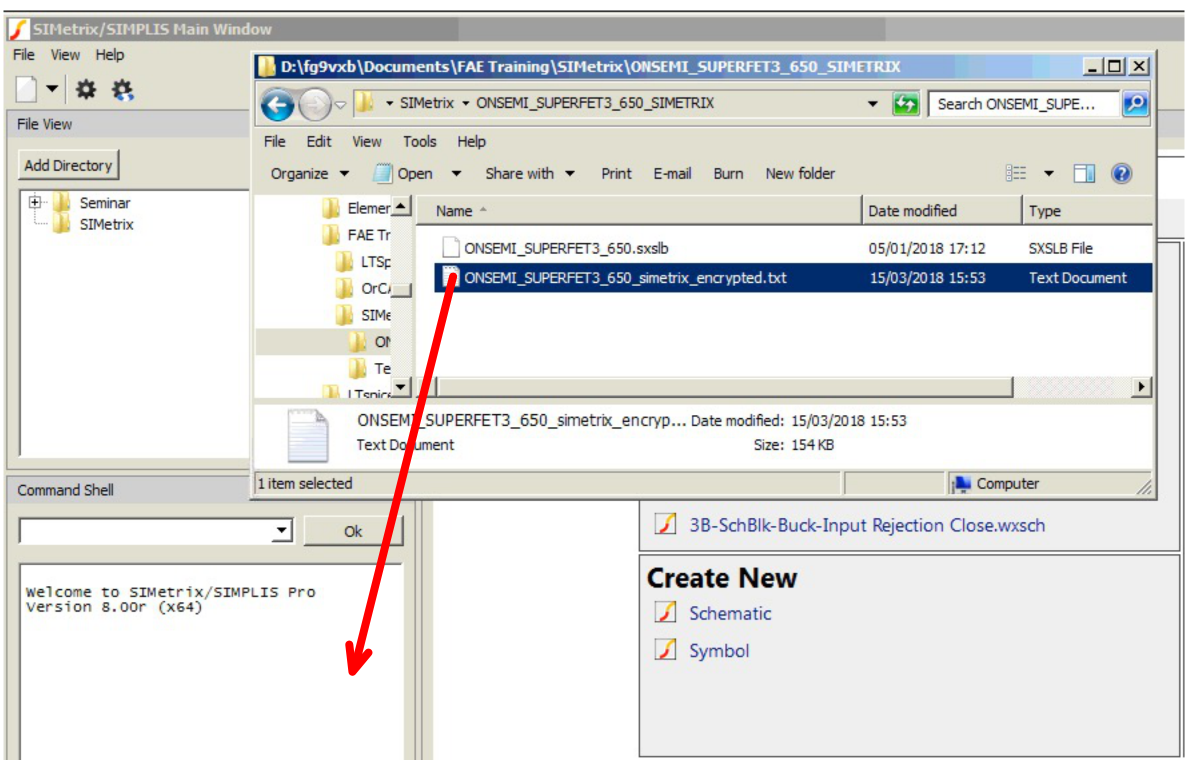


Figure 11. Importing the Sub−Circuits' Text File

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

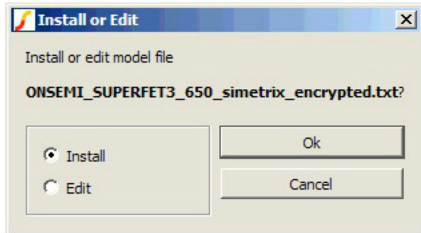


Figure 12. Importing Sub−Circuits Validation

Dragging the symbols’ file to SIMetrix

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

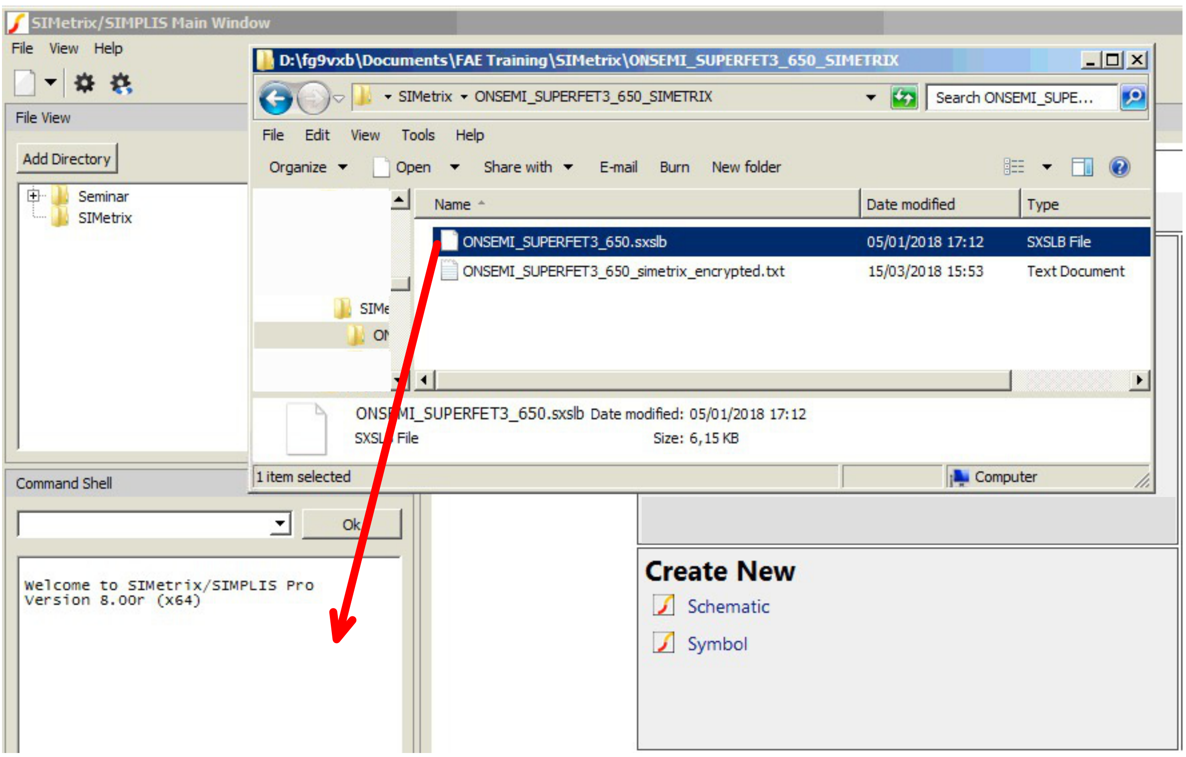


Figure 13. Importing the Symbols’ File

Then, SIMetrix asks what to do, select “OK”.

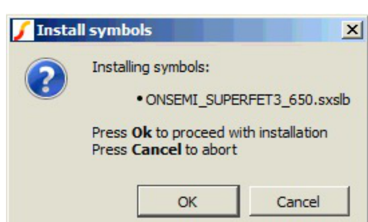


Figure 14. Importing Symbols Validation

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.

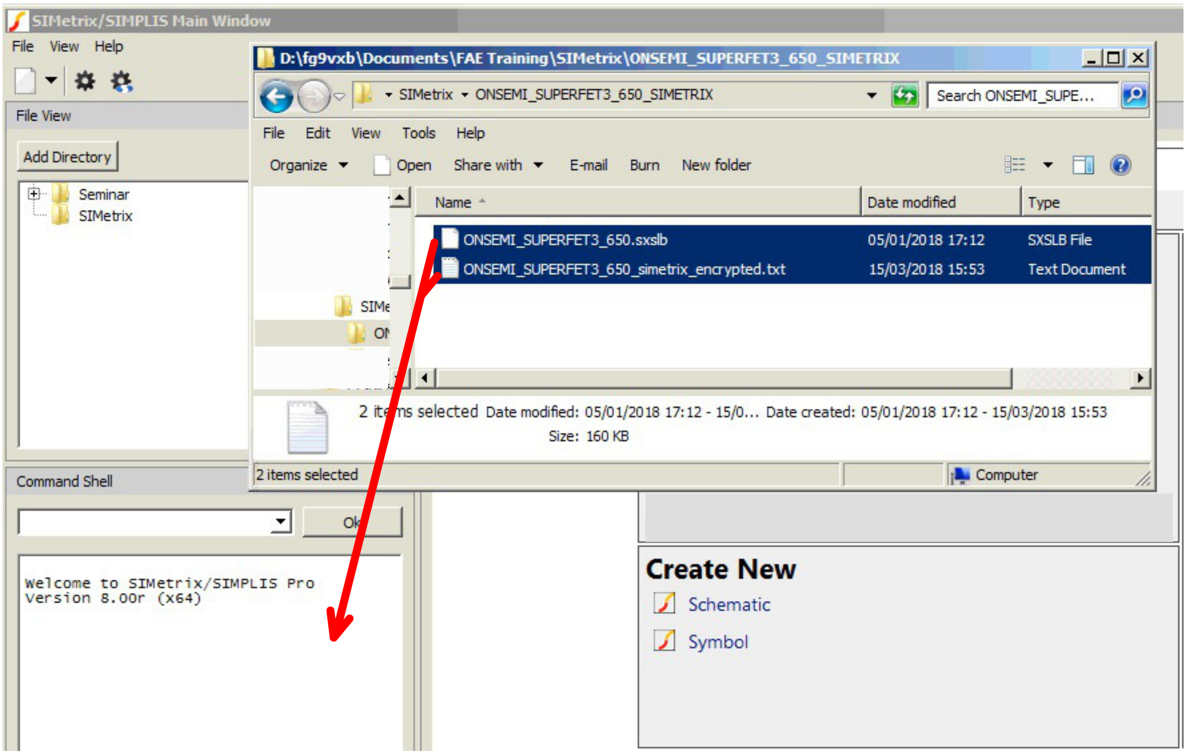


Figure 15. Importing the Sub–Circuits’ Text File and the Symbols File Together

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:

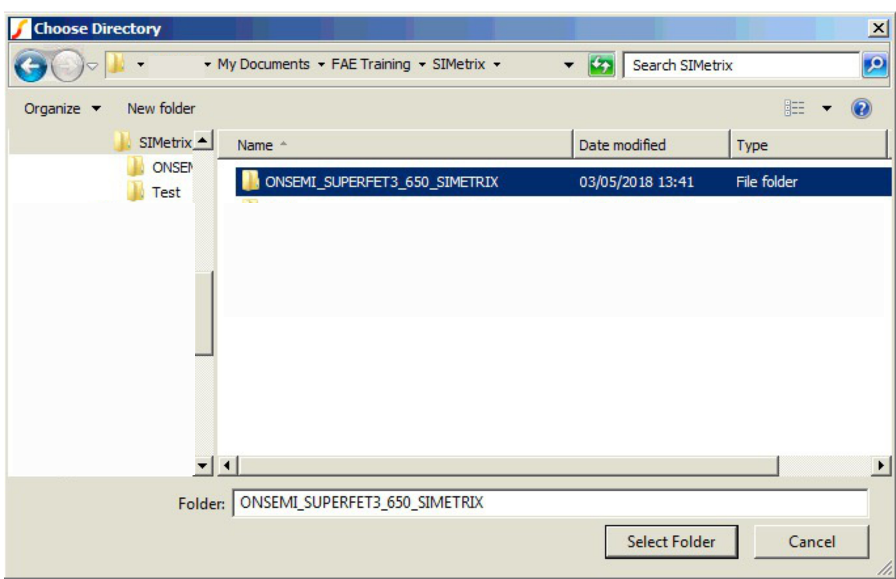


Figure 16. Selecting Sub–Circuits’ Text File Directory

The following sub−circuits’ library manager window opens with the library waiting to be added:

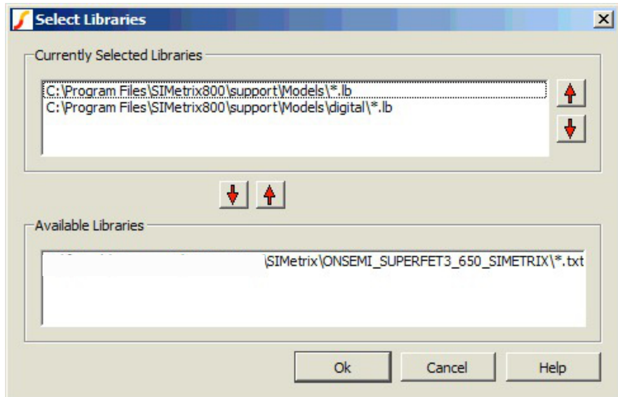


Figure 17. Sub−Circuits Library Waiting to be Included

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

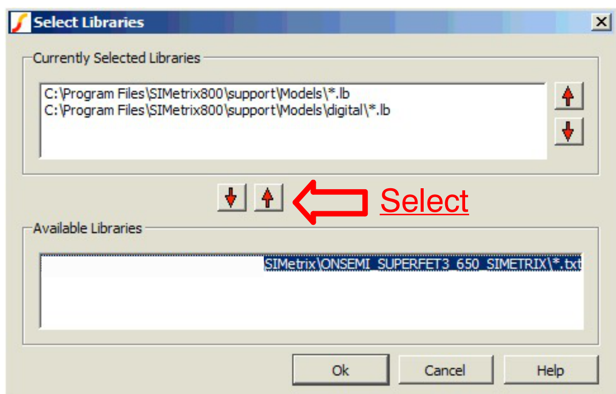


Figure 18. Library Selection and Inclusion

At the end, the sub−circuits’ library manager should look like this:

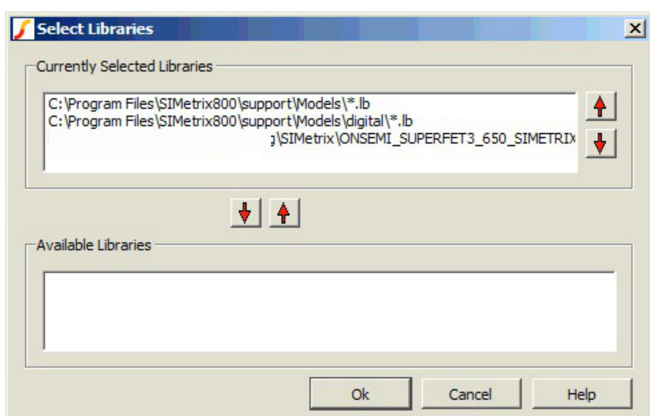


Figure 19. Sub−Circuits’ Library Text File Added

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:

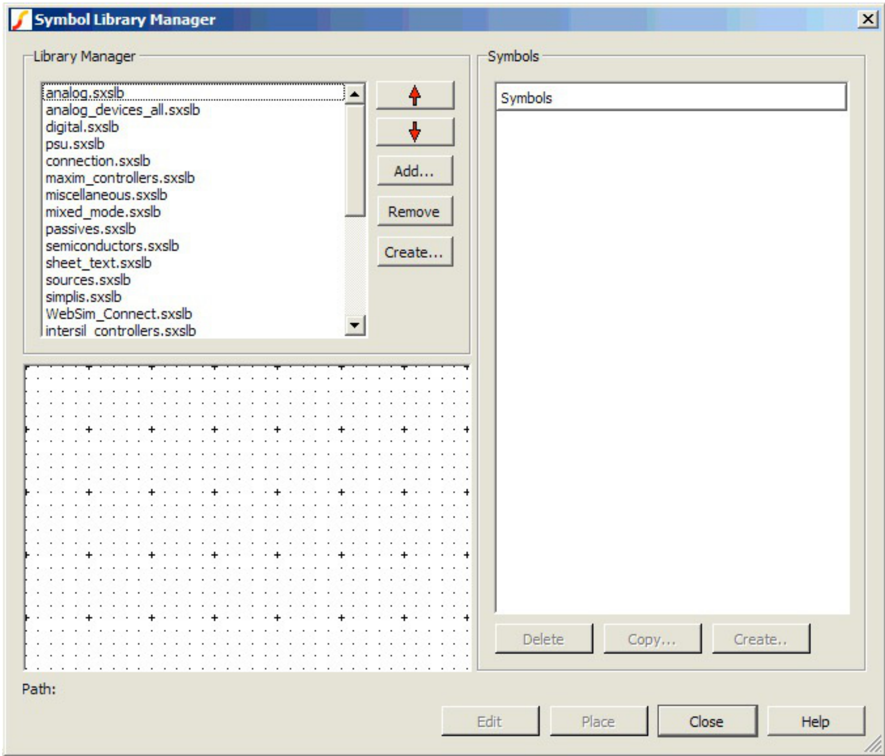


Figure 20. Symbol Manger to Include the Symbols’ Library

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

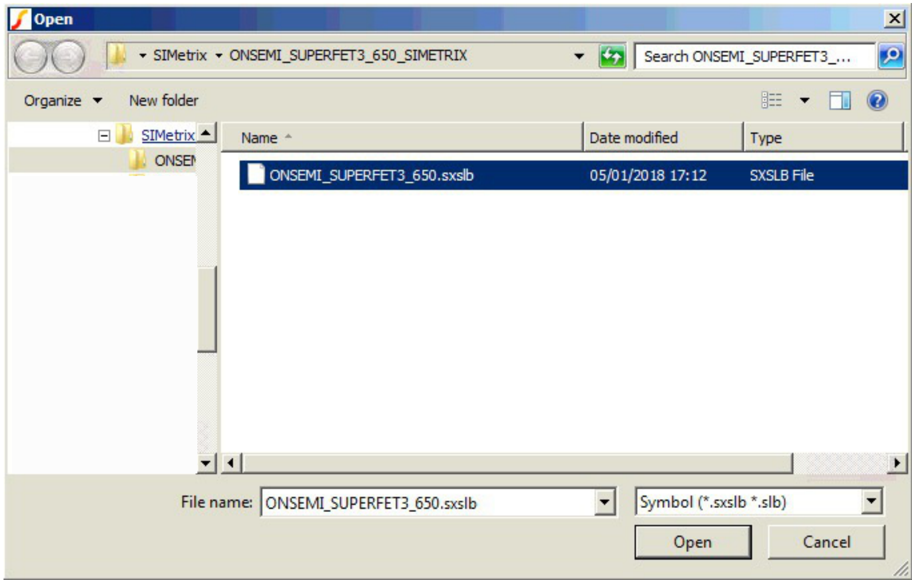


Figure 21. Selecting Symbols’ File

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

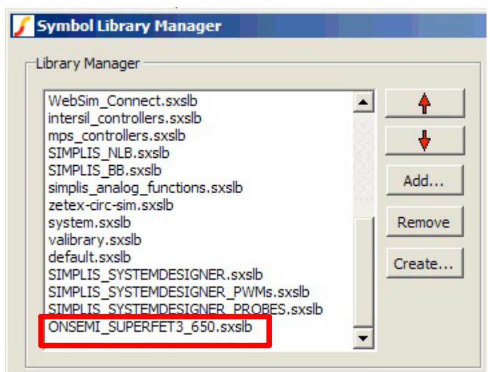


Figure 22. Symbols Added in the Symbols’ Library

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.

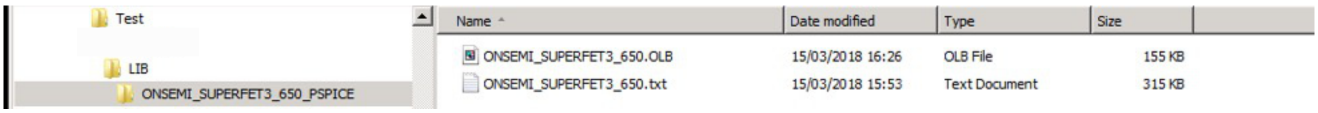


Figure 23. Unzipping the Library Files

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:

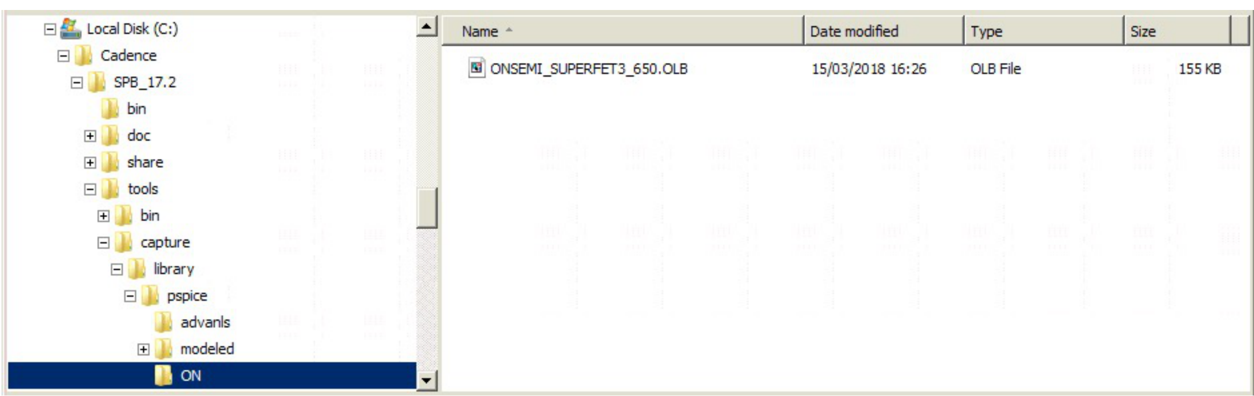


Figure 24. Copying ON Symbol Library File in OrCAD Capture Library Directory for Pspice

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:

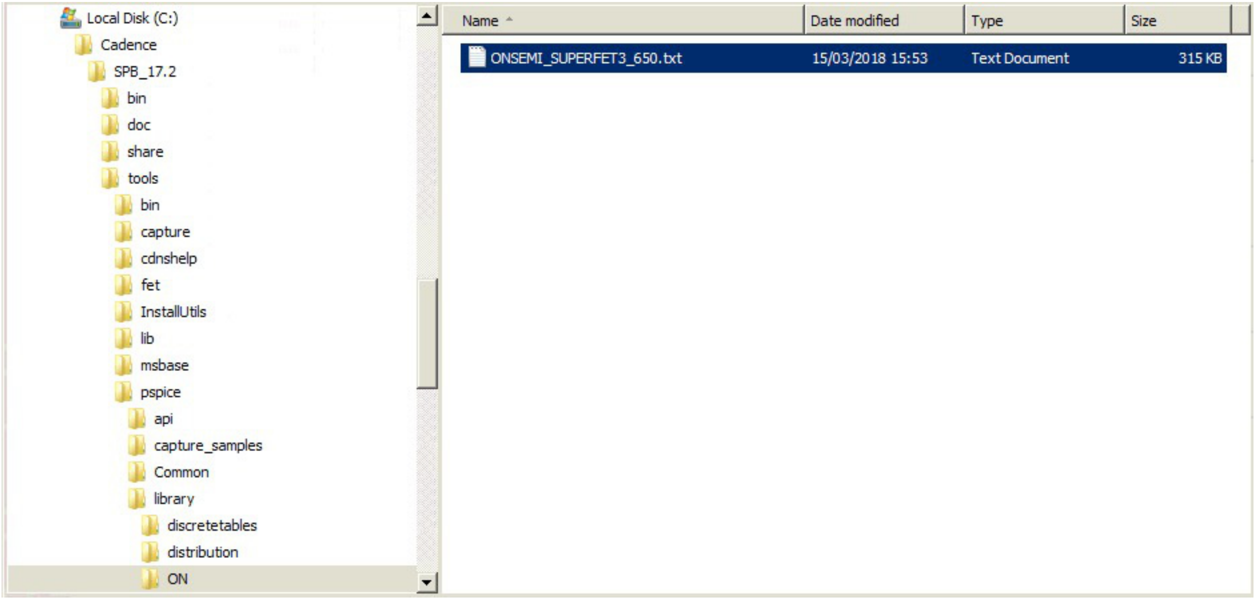
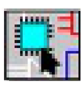


Figure 25. Copying ON Sub−Circuit Library File in OrCAD PSpice Library Directory

Launch OrCAD capture: 

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:

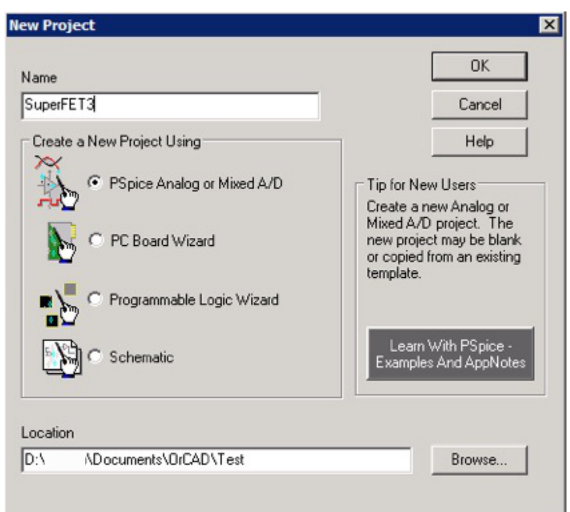


Figure 26. New Project Window Setup

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.

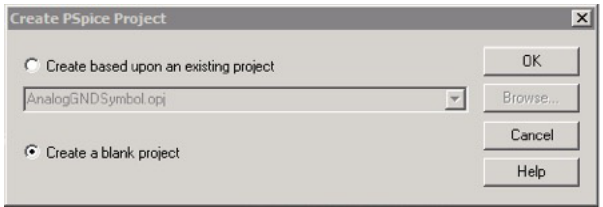


Figure 27. New Project Setup

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

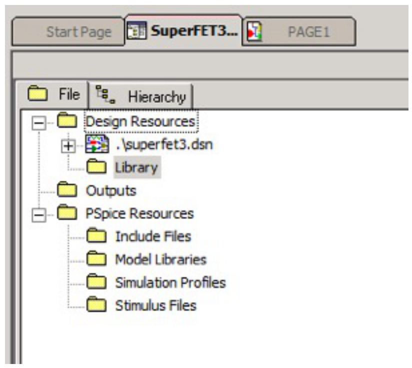


Figure 28. Complete Project Tree

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:

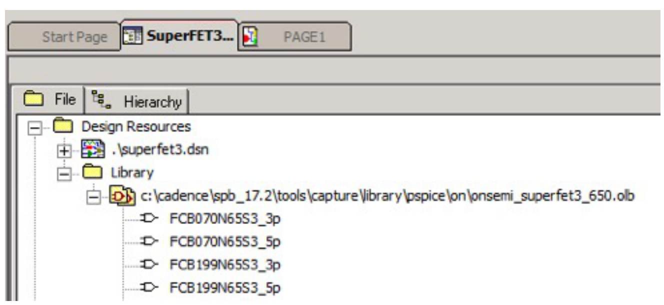


Figure 29. ON Symbol Library Loaded in the Project

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:

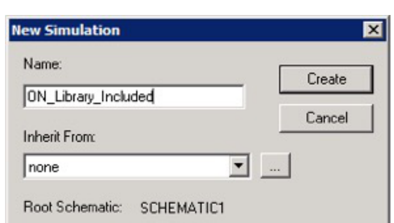


Figure 30. New Simulation Profile to Include Sub−Circuits’ Definitions

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

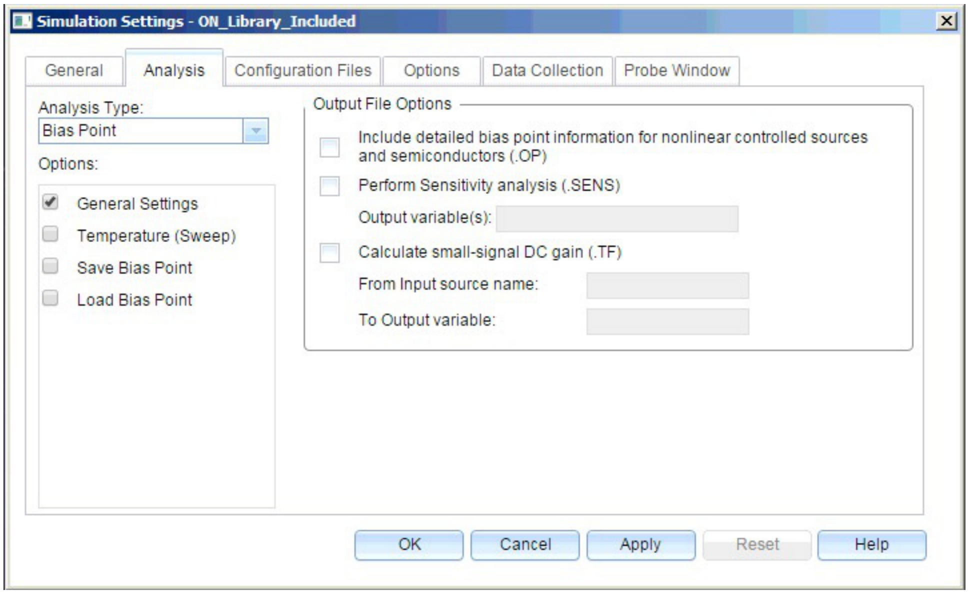


Figure 31. Bias Point Simulation as Default Profile

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

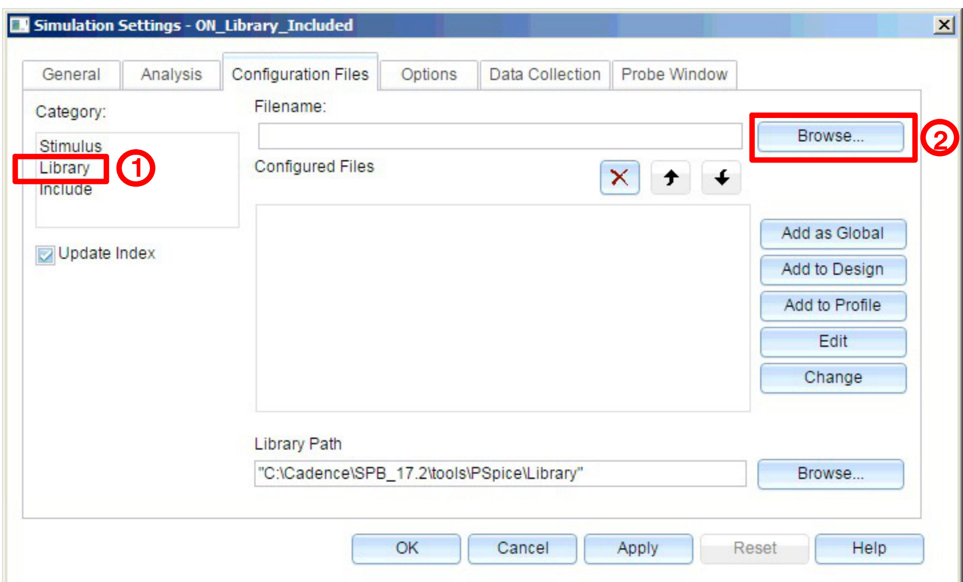


Figure 32. Simulation Setting Configuration Files Tab

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.

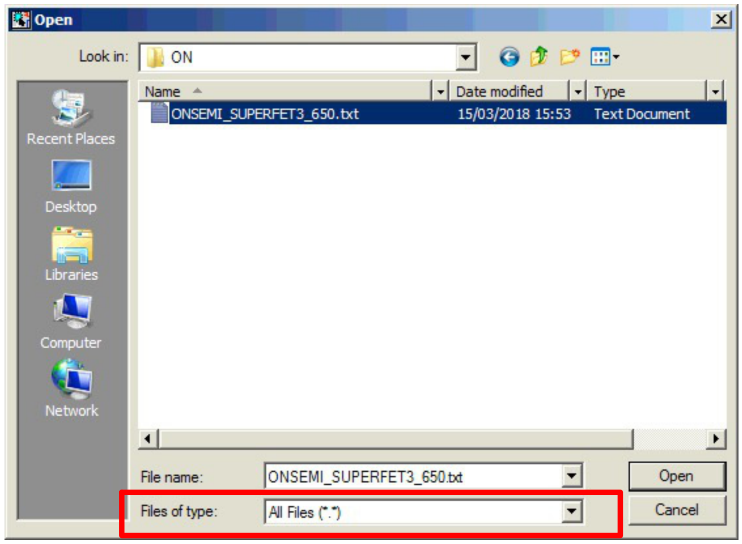


Figure 33. Selecting the Text File Containing the Sub−Circuits’ Definitions

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

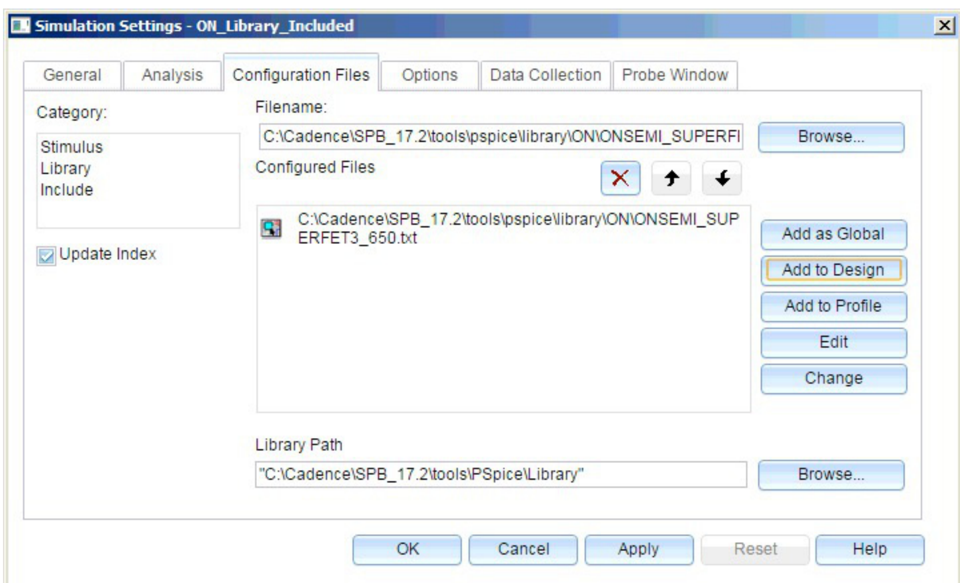


Figure 34. New Simulation Profile Setup With Sub−Circuits’ Definitions Included

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:

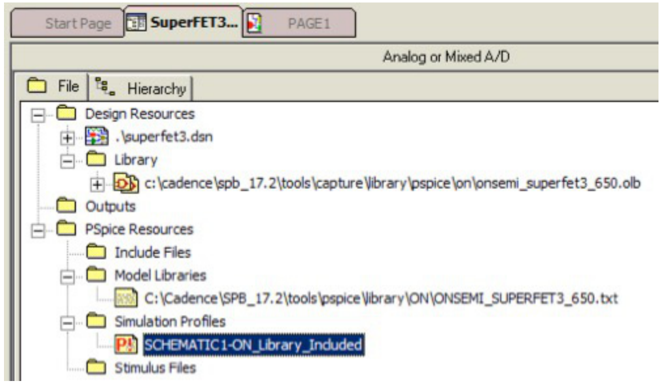


Figure 35. Project Tree With All ON Library Files

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:

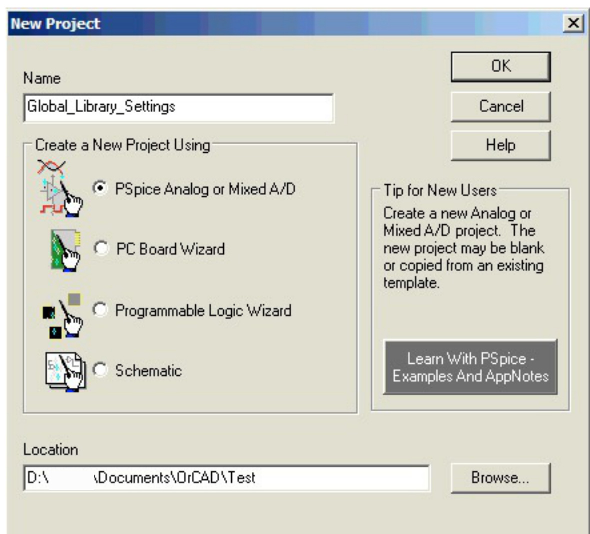


Figure 36. New Project Window Setup

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.

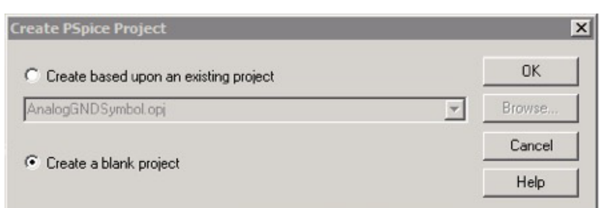


Figure 37. New Project Setup

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:

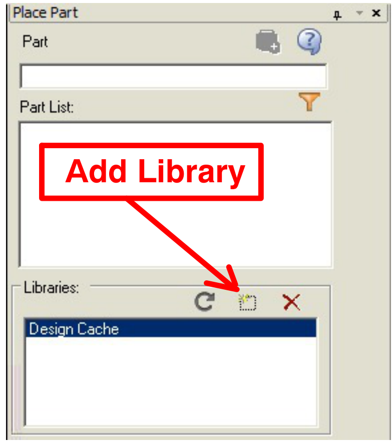


Figure 38. Add Library Button

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:

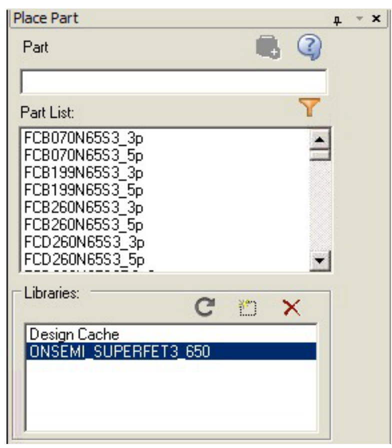


Figure 39. Symbol Library File Added as Global

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:

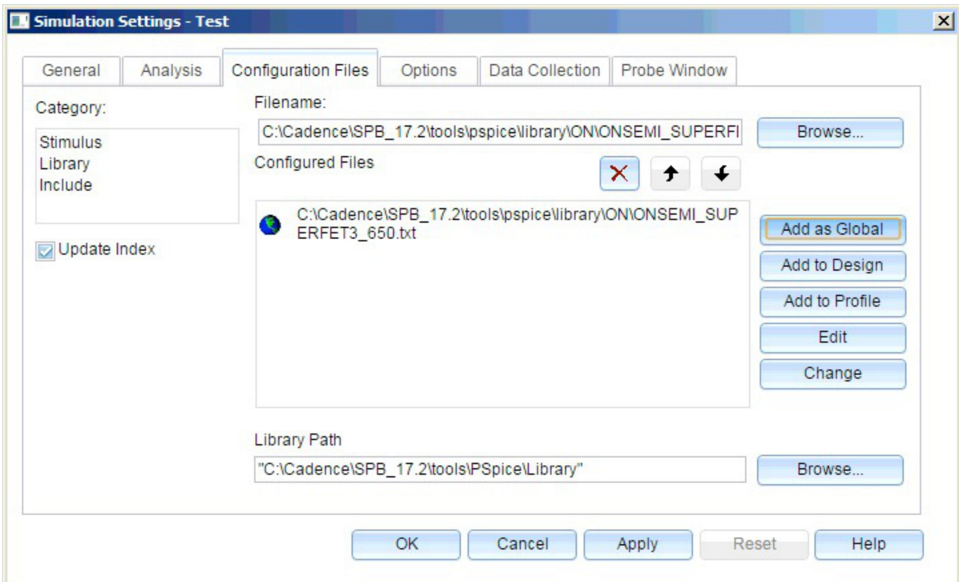


Figure 40. New Simulation Profile Setup With Sub−Circuits' Definitions Included

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.

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.

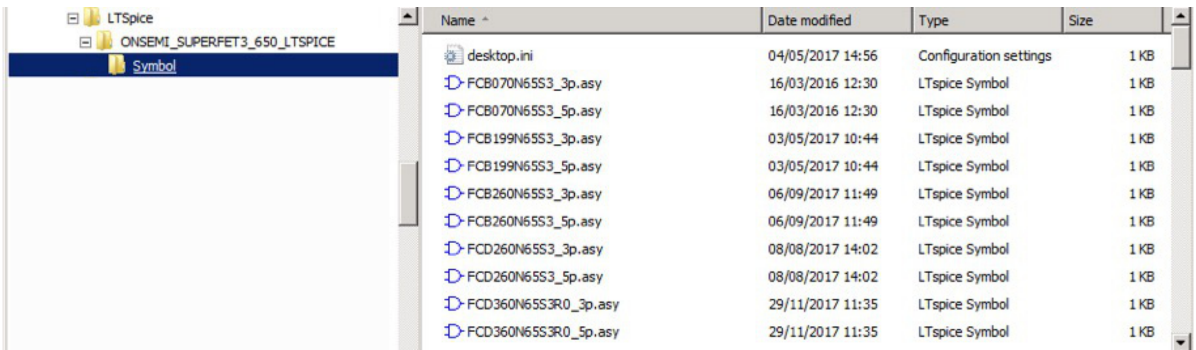


Figure 42. All Symbols Available

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.

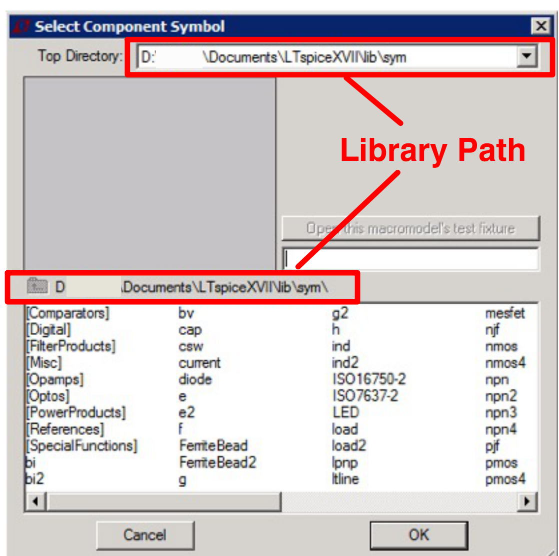


Figure 43. Library Path

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.

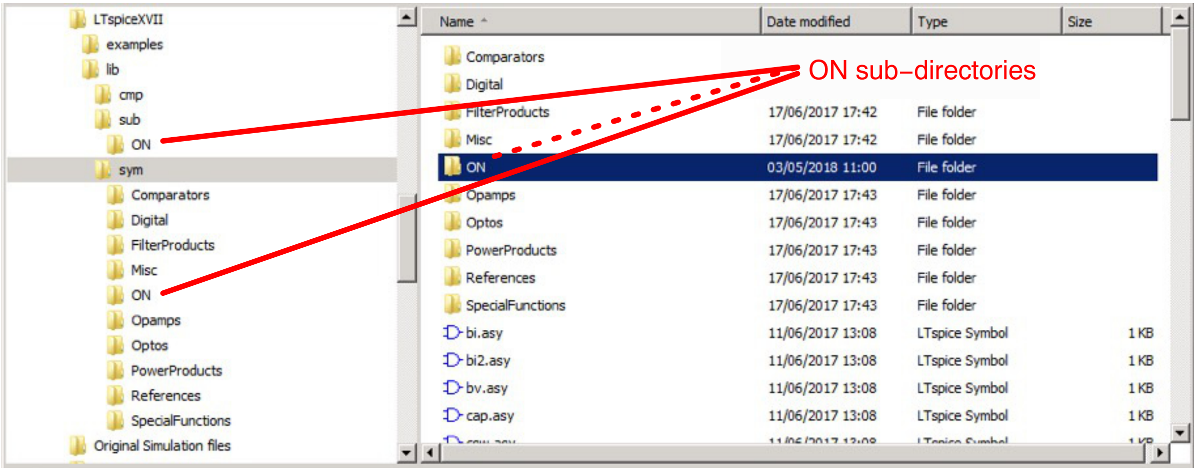


Figure 44. ON Sub−Directories for Sub−Circuit and Symbols' Files

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.

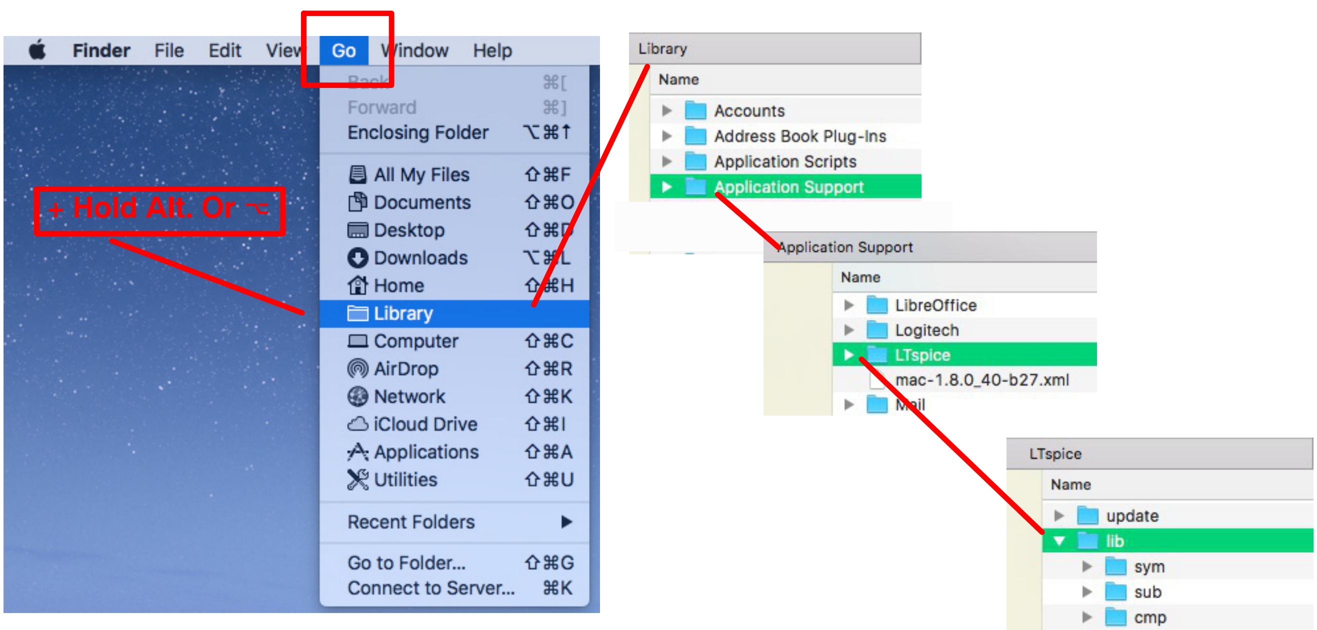


Figure 45. Library Folder Access on a MAC

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: and create a new schematic.

In the schematic, add a spice directive with the statement: ".Lib ON\LibraryF*ileName*.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.

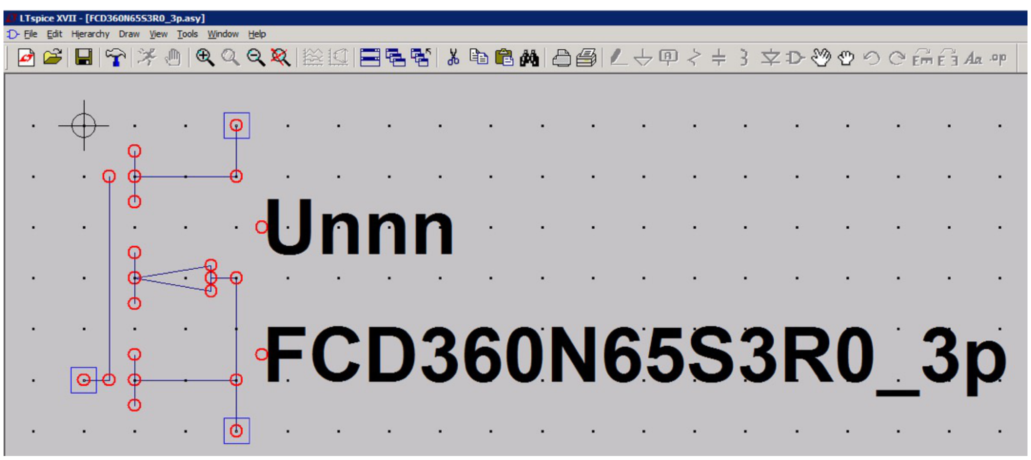


Figure 47. LTSpice Symbol Editor

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\Library*FileName*.txt”. See blow:

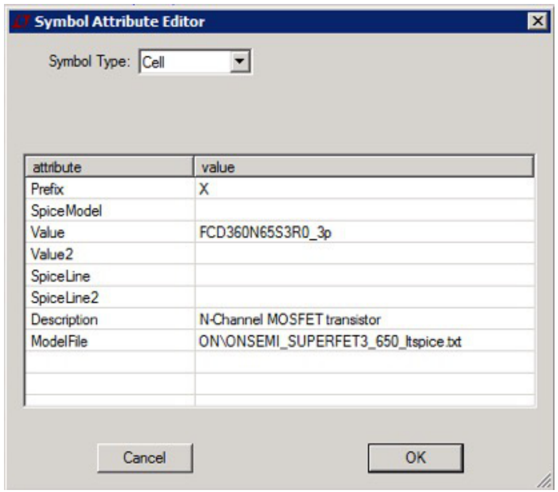


Figure 48. Add the Sub−Circuits’ Definitions’ File Path to the Symbol

Then, click OK, save the symbol. Quit and re−launch LTSpice: 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:

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

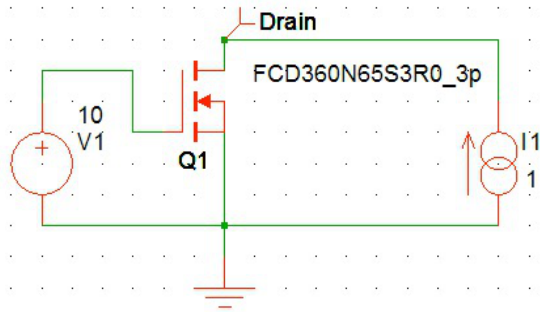


Figure 50. ON Resistance Measurement Setup With SIMetrix

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.

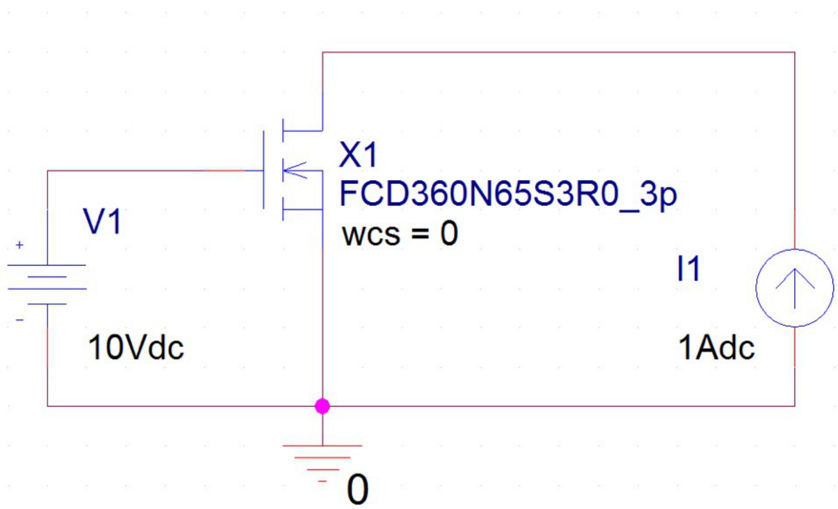


Figure 51. ON Resistance Measurement Setup With OrCAD Pspice

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.

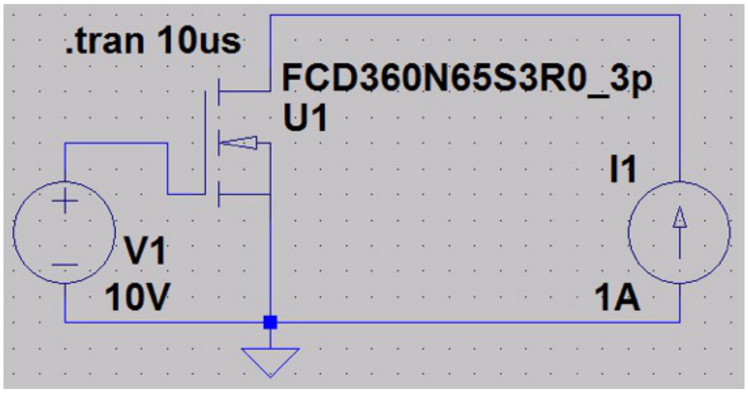


Figure 52. ON Resistance Measurement Setup With LTSpice

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.

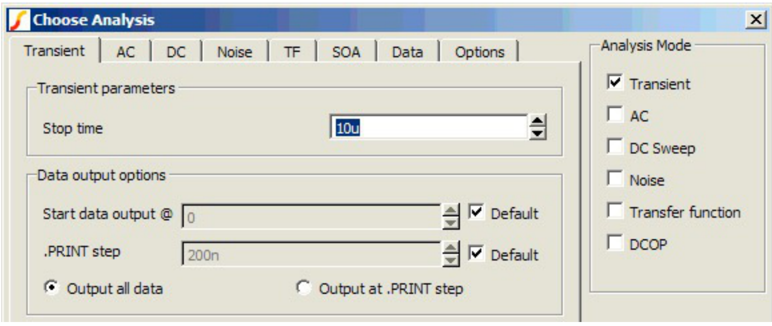


Figure 53. Simulation Analysis Selection With SIMetrix

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.

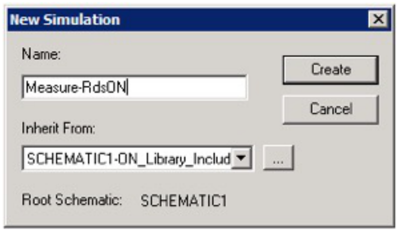


Figure 54. Inherit Simulation Profile Creation With OrCAD Pspice

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 –> Edit Simulation Cmd". Create a basic "Transient" simulation for 10 μs for example.

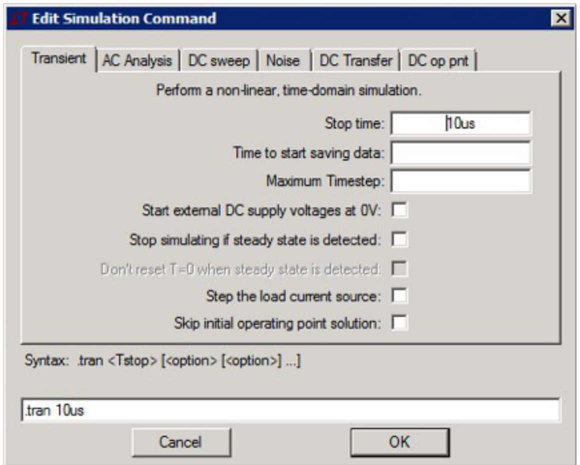


Figure 55. New Simulation Command Creation With LTSpice

Click Ok and place the statement in the schematic.

Results

With SIMetrix

Then, just run this new time domain simulation.

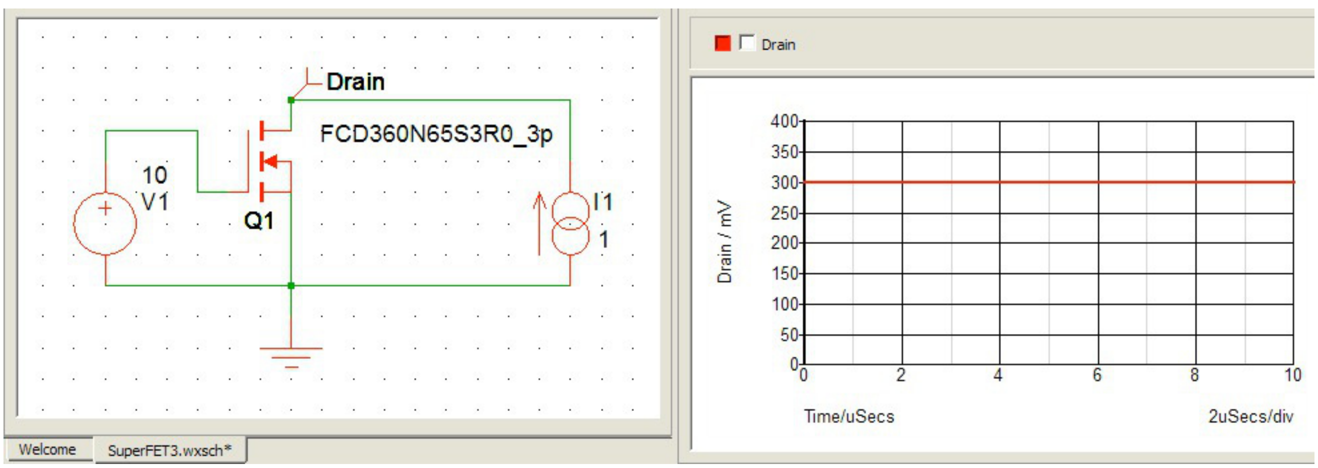


Figure 56. Simulation Results Obtained With SIMetrix

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

With OrCAD

Then, just run this new time domain simulation.

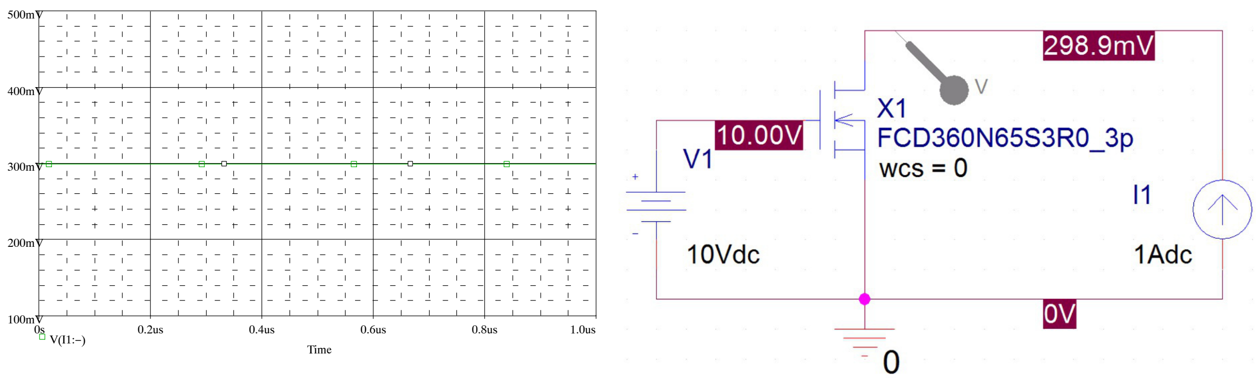


Figure 57. Simulation Results Obtained With OrCAD Pspice

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

With LTSpice

Then, just run this new time domain simulation.

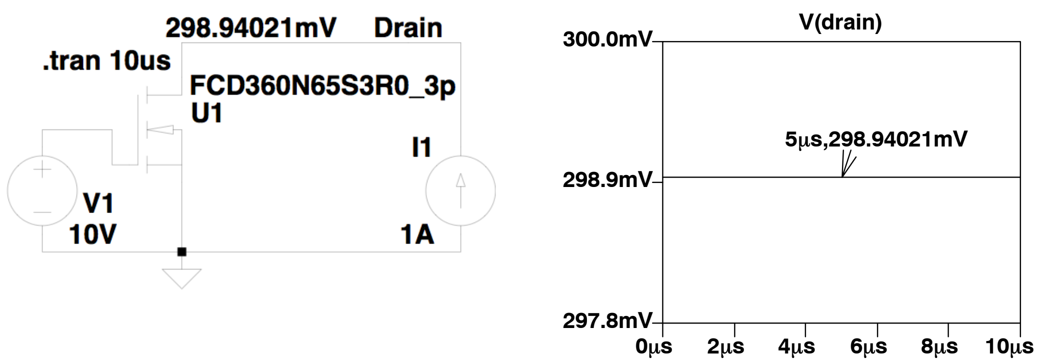


Figure 58. Simulation Results Obtained With LTSpice

Place a marker on the curve to measure a 298.9 mΩ 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

1. "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. http://www.orcad.com
4. http://www.analog.com/en/design−center/design−tools−and−calculators/ltspice−simulator.html

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