APPLICATION NOTE
www.onsemi.com

Usage of SIMetrix to Study MOSFETs Thermal Behaviors on Heatsink

Usage of Mixed−signal Circuit Simulator to Study Thermal Behavior on Power Components

AND90096/D

Prepared by: Benedicte Crosnier de Bellaistre
Applications Engineer

ABSTRACT

Thanks to the electrical analogy to thermal behaviors, we can use electronic simulators to build models of all scales of complexity to validate the thermal behavior of a die in its housing and on a heatsink.

This application note focuses on the use of available MOSFET Spice thermal models and heatsink Cauer networks to demonstrate how to build an easy to tune SIMerix electro−thermal simulation. Readers will then be able to use the same method on any electrical simulation using MOSFETs models with built−in thermal models.

Such simulation gives for instance the opportunity to check the impact of stress levels to the thermal evolution of a die in order to predict heat evolution throughout the die lifecycle.

INTRODUCTION

Thermal simulations are essential for the validation of discrete semiconductors.

As a mixed−signal circuit simulator, SIMetrix simulates electrical behaviors depending on a high variety of parameters, such as current or voltage waveforms and initial temperature. Electrical and thermal quantities have equivalent behaviors. Therefore, SIMetrix can also be used to simulate thermal responses using adequate transposition solutions. This paper describes how these simulations work, how to build them and what use cases can be studied using such simulation. The last section of this paper can also serve as a summary of the main thermal behaviors which can be expected from discrete components.

This paper will focus in particular on MOSFETs. However, this kind of thermal study can be applied to any similar semiconductor (IGBT, SiC transistor...).

ELECTRO−THERMAL SIMULATION PRINCIPLE

SIMetrix Presentation

SIMetrix is a mixed signal circuit simulator which can be used to run analog and power simulations. User can apply a variety of options and functionalities to meet their simulation needs, up to script creation.

Temperature management is accepted by SIMetrix through initial temperature set up. This temperature is relevant only for components which characteristics variation with temperature is integrated in the model.

Most of **onsemi** active discrete components are available on the **onsemi** website. These models are linked to temperature, see their presentation in section *“**onsemi** thermal models for discrete components presentation”*.

Thermal and Electrical Quantities Analogy and Thermal Behavior of Components

Using an electrical simulator may seem inadequate for an electro−thermal simulation. However, it is established that thermal and electrical quantities present enough analogies to model thermal behaviors through electrical circuits.

The table down below shows the relations between quantities important for our study.

Table 1. THERMAL AND ELECTRICAL QUANTITIES ANALOGY

Data	Thermal	Electrical
Potential	Temperature T [K]	Potential V [V = J/C]
Flux	Heat transfer rate dQ/dt [W = J/s]	Current I [A = C/s]
Resistance	Thermal resistance R [K/W]	Electrical resistance R [Ω]
Resistivity	Thermal resistivity R_λ [(m·K)/W]	Electrical resistivity ϱ [Ω·m]
Capacitance	Thermal capacity C_{th} [J/K]	Capacitance C [F]
Conductivity	Newton’s law of cooling $\Delta T = dQ/dt \cdot R$	Ohm’s law $\Delta V = IR$
	$dQ/dt = C_{th}\ \Delta T$	Capacitor behavior $I = C \cdot (dU/dt)$

In this table, the equation $\Delta T = dQ/dt \cdot R$ shows how temperature is linked to the electrical stress on the MOSFET.

Indeed, with the hypothesis that all the heat transfer rate for the MOSFET comes from Joule losses, this shows a direct relation between MOSFET temperature and the electrical stress it is put under.

Joules losses are transcribed as $P = R_{DS(ON)}\ I^2 + P_{commutation}$. With P the power through the MOSFET, $R_{DS(ON)}$ the ON resistance of the MOSFET and I the current through the MOSFET. This equation gives a direct relation between temperature and current for the MOSFET during MOSFET on time: $\Delta T = R_{DS(ON)}\ I^2 + R_{th}$

October, 2021 − Rev. 1

Since $R_{DS(ON)}$ varies according to temperature, the electro−thermal simulation offers a really good opportunity for easy automated temperature calculation.

MODELS PRESENTATION

onsemi Thermal Models for Discrete Components Presentation

Discrete semiconductors show variable characteristics with temperature.

To ensure the best accuracy while using models inside an electronic simulation, **onsemi** models are based on thermal behaviors of the components. These behaviors are then transcribed through curve fitting as a Cauer network (see the following "*Cauer network*" section for Cauer network description). **onsemi** electrical models of the MOSFETs are complex behavioral data.

For more details on how the models are built for MOSFETs, other documents by James Victory [1], [2], [3] are available.

These models are 5 pins models as presented in Figure 1.. They include a junction temperature reading *[Tj]* and a housing temperature connection *[Tcase]*, which should be linked to the thermal model of the substrate, usually a Cauer network to ensure models consistency.

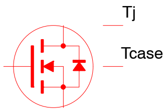


Figure 1. MOSFET Thermal Model

Cauer Network

Principle

To model the thermal behavior of a mechanical stack up (substrate and heatsink for instance, PCB in this paper), a Cauer network is used.

A Cauer network is originally a network to model the electrical behavior of a linear circuit. However, since there is an equivalence between electrical and thermal quantities, we can also built up thermal Cauer networks.

In networks modelling, two main methods can be found: Foster and Cauer. Foster networks are easy to build through curve−fitting of the thermal impedance of the component. Cauer models have a more complex expression, still obtained using curve−fitting, but their structure respects the mechanical stack−up of components.

This is why Cauer networks are used in the silicon industry to create thermal models for substrates, heatsinks and discrete components. Since the model corresponds to the physical stack−up, two different models can be linked by a simple temperature connection.

How to Build it

To get the Cauer network model for any mechanical stack−up, including MOSFET housing or heatsink, one needs to follow three steps and use specific software:

- Obtain a precise and complete 3D model of the heatsink, including precise description of the contact area with the power component studied.
- Create the thermal impedance (Zth) curve using a calculator and measurements to fit curves. Well−known Computational Fluid Dynamics (CFD) software packages allow users to run thermal impedance simulation.
- Compute the Cauer network (usually included in CFD software)

This application note is provided with examples of PCB described in section "*Example of simulation definition*" and annex "*Cauer models for simulations*".

Schematics Description

A complete and precise Cauer network for a mechanical stack up should be as follow:

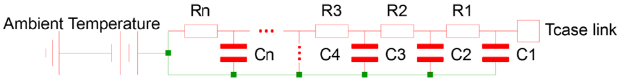


Figure 2. Cauer Network

- [*Capacitors*] model the thermal capacity of the component. It is linked to both the ambient temperature and the component since the heat comes from these two sources.
- [*Resistances*] model the thermal resistivity of the components. It is the image of temperature gradient across the component.
- [*Tcaselink*] is the SIMetrix component for voltage links in the simulation. (Component to be found in "Connectors" → "terminal")
- [*Power supply source*] model ambient temperature as expressed. Using this voltage source means the ambient temperature works as a perfect heat source which cannot change. This is a common assumption for power electronics situations. It is important to define the right iso−temperature of the application beforehand. For instance, while studying a power component in a full alternator build, the ambient temperature might not be immediate surrounding of the component. It might be instead the outer protection housing, or even the air outside of the alternator. This depends on temperature evolution during the application. The ambient temperature modeled with a constant voltage source must be in fact constant during the time of study of the application. If not, two solutions can be chosen:
    1. Build a more complete Cauer network to reach another iso−temperature.
    2. If the temperature of the "ambient temperature" considered evolves with no correlation with the one of the studied component (for instance, heating throughout the day of an outside set up): Use a

measurement or other type of simulation to know the evolution of the “ambient temperature”. A function can be built in SIMetrix to model this evolution as shown down below:

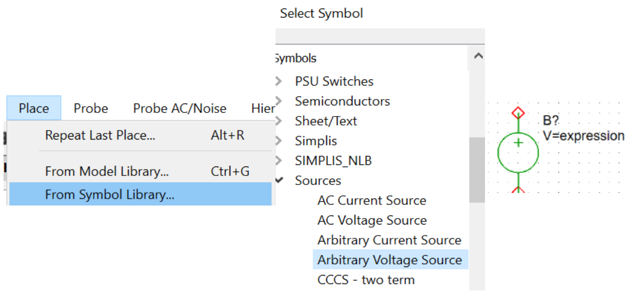


Figure 3. Arbitrary Voltage Source Access

The expression is a function of time, using the parameter “time” precisely. For instance: *V = 3 * time*

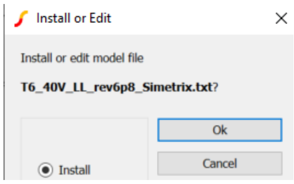
BUILD A THERMAL SIMULATION

This application note focuses on describing the process to build a simulation in SIMetrix. The basics are the same for other software, user can see paper [5] for data on the subject.

How to Download Components from onsemi Website

On the **onsemi** website, on the page of a component, there is a “technical documentation & design resources” section. There should be SPICE/SIMetrix models for the component. (For the complete procedure, refer to application note [5]) If not, users can always contact **onsemi** to get missing data.

After the download, the component can be integrated to SIMetrix using a drag and drop. The model file (and, if needed, the symbol file) needs to be dragged and dropped in the “part selector” section of SIMetrix. If the process worked, a pop up should appear:



Install or edit model file

T6_40V_LL_rev6p8_Simetrix.txt?

Figure 4. Install Pop Up

After accepting install, the component can be found in the “recently added models” section and used directly.

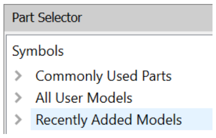


Figure 5. New Model Access

It is important to know that this procedure adds the model to the personal library of the user. Therefore, if the simulation is used on a different computer, the component needs to be added again to the library. It may appear on the simulation schematics, but when the simulation is run, a warning for “not found component” will be displayed in the command shell.

Example of Simulation Definition

The following simulation is specifically set up to allow a thermal study of the component on a specific substrate.

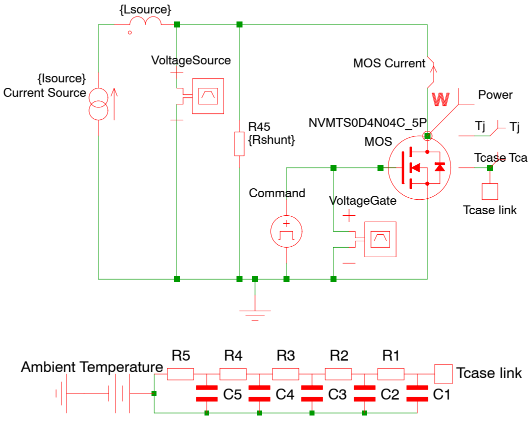


Figure 6. Simulation Schematics

Components Description

Source *[Current Source]* – A current source is used here to ensure a constant current through the MOSFET.

Inductance *[Lsource]* – We use an inductance to ensure convergence in the simulation.

Resistance *[rshunt]* – Resistance ensures the MOSFET does not run into avalanche. If the voltage of the MOSFET reaches the maximum value allowed by the user, the resistance shunts the current. This shunt effect offers an easy way to detect avalanche mode and shut down the stress on the MOSFET.

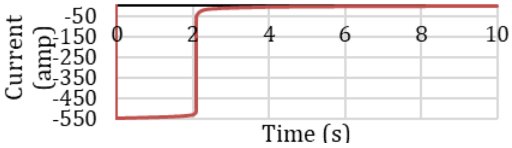


Figure 7. Resistance Effect Simulation Result

Cauer Network − MOSFET Cauer network is directly integrated in MOSFETs models as described in section “*ON Semiconductor Thermal Models for Discrete Components Presentation*”.

The Cauer network used in the simulation is to model the PCB. It is based on a 1s0p PCB (one layer) with different copper thickness and area size. The network was calculated as a Foster network. A transfer matrix was used to convert this Foster to a Cauer form. Although the network is an accurate representation of the complete PCB thermal stack−up, it cannot be divided along the mechanical stack−up elements as described in section “*Cauer Network*”.

Figure 8 represents the full thermal schematics in this simulation, with both the MOSFET internal thermal network (as shown in sketch, included in MOSFET 5 pins model) and the PCB thermal model represented:

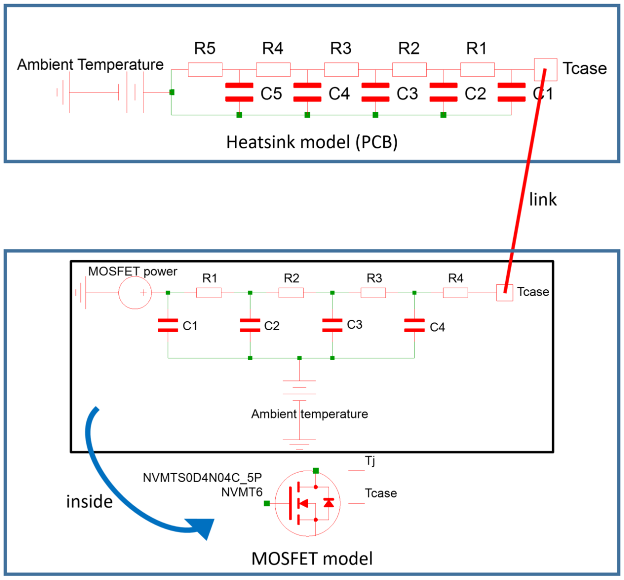


Figure 8. MOSFET and PCB Thermal Models

In SIMetrix, it is possible to duplicate the schematics to have a simulation set up per Cauer network used on the same schematics sheet. However, this brings a rather difficult to read and modify simulation.

The simulation available with this application note is therefore built differently. A script is used to automatically modify the Cauer network on the single schematics simulation, and run the simulation for each new Cauer network.

This script is based on SIMetrix help page [4] and is quite simple to reproduce. The script with annotations to navigate is added in annex “*Script to run several simulations*”.

A text file is needed to provide the Cauer networks Capacitors and Resistances values. This text file must be designed as follow (a design in a spreadsheet copied and pasted in a notepad application works perfectly fine):

- First line: R and C names, the same as the ones in the simulation
- Second line: corresponding values

For instance, for a 2 stacks Cauer network (containing therefore 2 R and 2 C), we could have the following array:

R1	R2	C1	C2
4.5	10	120m	150m

The order of the R and Cs is not important for the script. It is simply to make notepad writing easier.

Command − Several commands will be used in the presentation to achieve different functions.

- Waveform generator

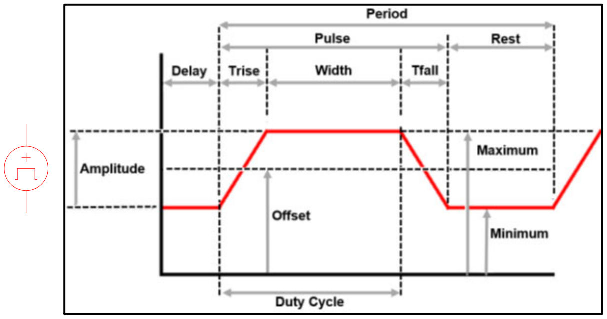


Figure 9. Waveform Symbol and Definition

The waveform generator is very useful to create simple MOSFET commands, with tuned on and off times.

This periodic waveform can be used as a single pulse with the following set up:

$Width = Pulse\ Duration - T_{rise} - T_{fall}$

$Simulation\ Duration = Period$

$= Pulse\ Duration + Rest\ Duration$

- Volatge source set on “pulse” mode

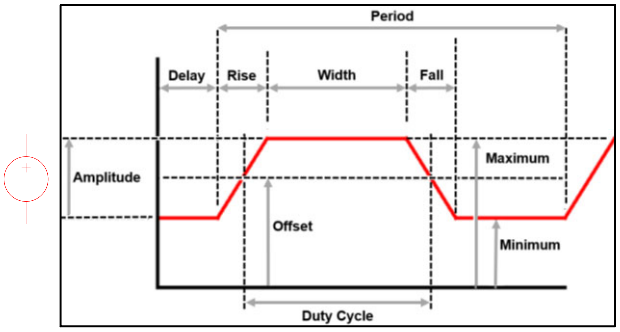


Figure 10. Waveform Symbol and Definition

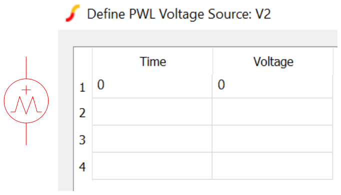
This voltage source is basically the same as the waveform. The two main differences are the way the duty cycle is calculated, and the computing linked to this source.

The duty cycle is calculated as so

$Duty\ Cycle * Period = (T_{rise} / 2)\ Width\ (T_{fall} / 2)$

The computing using this source brings less convergence errors.

- PWL Source


Define PWL Voltage Source: V2

	Time	Voltage
1	0	0
2		
3		
4		

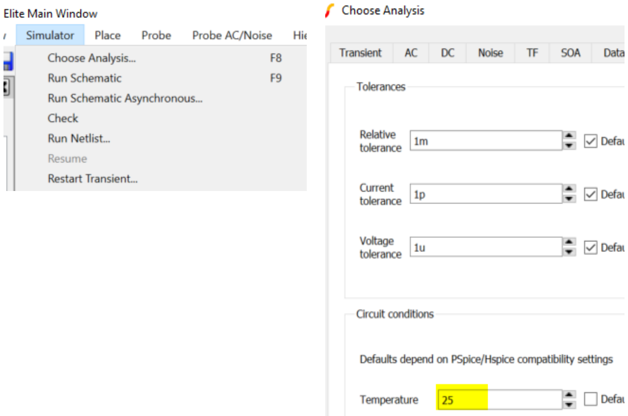
Figure 11. PWL Source Symbol and Definition

This voltage source allows the user to build a wave form from scratch. This is specifically useful for non−classical command (square, triangle, sinus, step).

Initial Temperature Set−up

Working with temperature in SIMetrix implies to carefully set up the initial temperature of the study. Two steps are required to do so.

First, set up the initial temperature in the simulation option:



Tolerances

Relative tolerance: 1m ☑ Defau
Current tolerance: 1p ☑ Defau
Voltage tolerance: 1u ☑ Defau

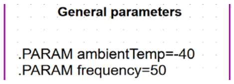
Circuit conditions

Defaults depend on PSpice/Hspice compatibility settings

Temperature: 25 ☐ Defau

Figure 12. Initial Temperature Set Up

Then, set up the ambient temperature in the schematics:

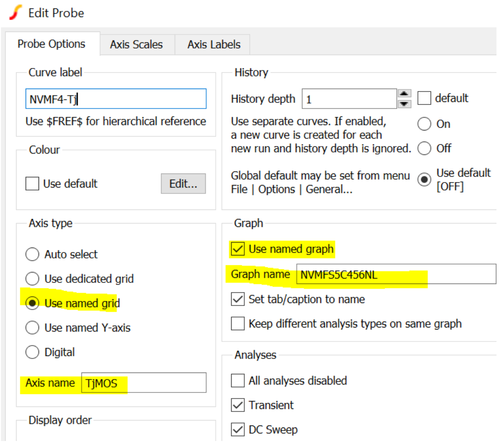

General parameters

.PARAM ambientTemp=-40
.PARAM frequency=50

Figure 13. Ambient Temperature Definition in Simulation

These two steps are both of equal importance: the first step ensures the models linked to temperature (such as MOSFET model) start at the right temperature. The second step ensures the other components linked to a temperature in the simulation (such as Cauer network for PCB) start also at the right temperature.

Probes Set−up



Curve label: NVMF4-Tj
Use $FREF$ for hierarchical reference

Colour
☐ Use default

Axis type
○ Auto select
○ Use dedicated grid
● Use named grid
○ Use named Y-axis
○ Digital
Axis name: TJMOS

Display order

History
History depth: 1 ☐ default
Use separate curves. If enabled, a new curve is created for each new run and history depth is ignored.
○ On
○ Off
Global default may be set from menu File | Options | General...
● Use default [OFF]

Graph
☑ Use named graph
Graph name: NVMFS5C456NL
☑ Set tab/caption to name
☐ Keep different analysis types on same graph

Analyses
☐ All analyses disabled
☑ Transient
☑ DC Sweep

Figure 14. Probe Definition

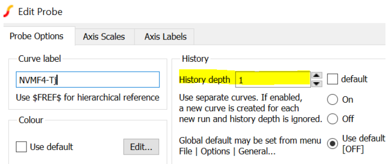
All the probes of the simulation are set up with specific elements to ensure a clean graph tab configuration.

“Axis type” is set up to have a different named grid for voltage, junction temperature, housing temperature… The name of the grid needs to be defined as well in “axis name” to ensure an easy understanding.

The “graph” section allows to have a new graph tab. It is important to use it if a lot of different curves are displayed. Change graph name to ensure a good understanding of what tab contains what curve.

Depending on whether the simulation is running with the script or with a repetition of the same schematics, two different set−ups for probes need to be used:

- For a regular (non−script) simulation, history depth needs to be set to 1. This parameter represents the number of time the curve for this probe will be kept after a new simulation. That is two say, if the history depth is set to 2, the first simulation after opening the graph page will be displayed. If a new simulation with different parameters is run, it will be displayed on the same graphs, with the previous curve still displayed. If yet another simulation is run, the first curve will disappear, and the graph will show only the two most recent curves.



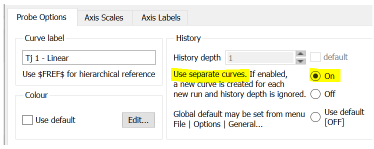
Curve label: NVMF4-Tj
Use $FREF$ for hierarchical reference

Colour
☐ Use default

History
History depth: 1 ☐ default
Use separate curves. If enabled, a new curve is created for each new run and history depth is ignored.
○ On
○ Off
Global default may be set from menu File | Options | General...
● Use default [OFF]

Figure 15. Regular Non−script Simulation

- For simulation using the script, it is really important to set the probe to “use separated curves”. This allows any new runs in the script to be saved and displayed.



Curve label: Tj 1 - Linear

Use $FREF$ for hierarchical reference

History

History depth 1 ☐ default

Use separate curves. If enabled, a new curve is created for each new run and history depth is ignored. ● On ○ Off

Global default may be set from menu File | Options | General... ○ Use default [OFF]

Colour: ☐ Use default

Figure 16. Scripted Simulation

Maximum Allowed Temperature Set Up

To see on the graphs the datasheet temperature range, a simple additional schematics can be added, with the appropriate settings.

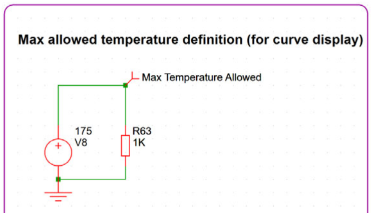
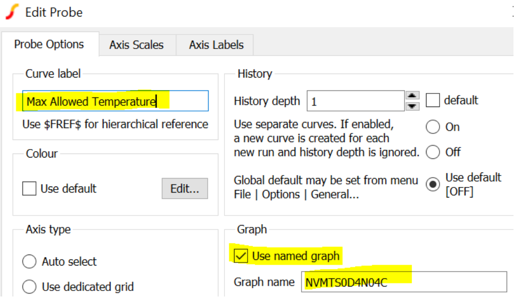


Figure 17. Max Temperature Schematics

The voltage source sets the maximum allowed temperature for the study.

The probe should be set to have the graph appear in the wanted tab as shown before:



Curve label: Max Allowed Temperature

Use $FREF$ for hierarchical reference

History

History depth 1 ☐ default

Use separate curves. If enabled, a new curve is created for each new run and history depth is ignored. ○ On ○ Off

Global default may be set from menu File | Options | General... ● Use default [OFF]

Colour: ☐ Use default

Axis type: ○ Auto select ○ Use dedicated grid

Graph: ☑ Use named graph

Graph name: NVMTS0D4N04C

Figure 18. Probe Definition for Max Temperature Display

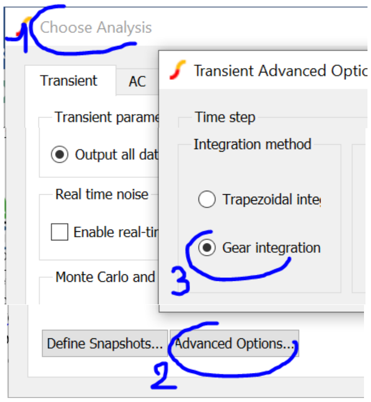
Common Errors from SIMetrix

**** ERROR *** No Convergence in Transient Analysis*

This error may be due to a too heavy simulation to run, or too many curves for the graphic processor.

Certain types of source may cause more calculation for the same result with SIMetrix. Depending on your needs, consider to always use a regular “voltage source” and not a “waveform source”.

As a last resource with simulations using complex command, the settings of the simulation can also be changed:

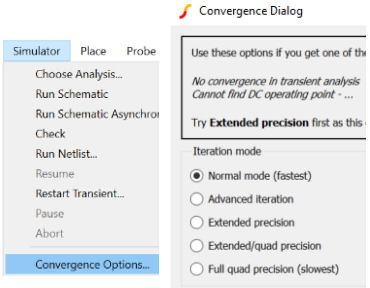


○ Trapezoidal inte

● Gear integration

Figure 19. Advanced Calculation Method Tab

- In the “choose analysis” tab, you can find in advanced options the integration method. This will slow down slightly the computation time but should solve any minor convergence issue.



Use these options if you get one of th

No convergence in transient analysis
Cannot find DC operating point - ...

Try **Extended precision** first as this

Iteration mode

● Normal mode (fastest)
○ Advanced iteration
○ Extended precision
○ Extended/quad precision
○ Full quad precision (slowest)

Figure 20. Advanced Convergence Options Tab

- For extended control over the computing method, the convergence option can be changed. The change tab for this option is really straightforward. As said in SIMetrix, the methods are slower and slower. However, if this allows the simulation to run without convergence errors, this is definitely a change to make.

**** ERROR *** Singular Matrix – This May be Due to a Floating Node or a Loop of Voltage Sources and/or Inductors.*

Two main reasons for this error:

1. Indeed an issue with a floating node; Understand that a node is not connected to the ground. To avoid this issue, connect a 1 MΩ resistor to willingly not connected nodes.
2. Loop of voltage source/inductors often happens when using LC models from 3D quasi static field solver. This issue can be solved by putting low inductances on outputs.

STUDY OF THERMAL EVOLUTION ON MOSFET

Important Information

When studying thermal behaviors on components, one should always unsure the results stay inside the application range of the component. Extrapolation may not be relevant for models.

Component Definition for the Study

For this simulation, two MOSFETs are used:

1. One with a lower $R_{DS(ON)}$ and a bigger housing:

Table 2. FIRST MOSFET MAIN POWER CHARACTERISTICS

MOS Reference	Characteristics			Housing
NVMTS0D4N04CL	$V_{(BR)DSS}$	$R_{DS(ON)}MAX$	I_DMAX	8x8 mm
	40 V	0.45 mΩ @ 10 V	550 A	

2. One with a higher $R_{DS(ON)}$ and a smaller housing:

Table 3. SECOND MOSFET MAIN POWER CHARACTERISTICS

MOS Reference	Characteristics			Housing
NVMFS5C456NL	$V_{(BR)DSS}$	$R_{DS(ON)}MAX$	I_DMAX	5x6 mm
	40 V	3.7 mΩ @ 10 V	87 A	
		6.0 mΩ @ 4.5 V		

Shortened names for the MOSFETs will be used throughout the document to simplify the reading and curves labels: NVMT for NVMTS0D4N04CL and NVMF for NVMFS5C456NL.

Four combinations of PCB definitions are used in the following studies. The corresponding Cauer networks R and C are available in annex "*Cauer models for simulations*". All PCB used are 1s0p PCBs. The four combinations goes as follow (letter codes are given for the curves found later on in this paper):

Table 4. PCB CONFIGURATION

MOS Housing Contact Area	Lower Copper Thickness	Higher Copper Thickness
Smaller Copper Area	LS	HS
Bigger Copper Area	LB	HB

- Lower Copper thickness corresponds to 1 oz/ft^2 copper trace thickness
- Higher Copper thickness corresponds to 2 oz/ft^2 copper trace thickness
- Smaller Copper area corresponds to 350 mm^2 copper area
- Higher Copper area corresponds to 800 mm^2 copper area

PCB Variations to Fit Thermal Stress

This section will demonstrate the impact the substrate and heatsink used to assemble a component have on the thermal behavior of this component.

The studies in this section will show the thermal evolutions depending on specific PCB characteristics described previously. The same method can be applied to study other heatsink changes, using the corresponding Cauer network.

Full set−up description:

The current used in this section is a continuous current of 90 amps. As explained in section I.B, temperature and current are linked for MOSFETs: The higher the current through the MOS, the higher the temperature.

• Frequency	0.001 Hz
• Current	90 amps
• Duty Cycle	99%
• Trise = Tfall	1 ns
• Initial temperature	25°C

Note that the frequency is low enough to have a continuous study over a few seconds.

Results Summary:

The result of this simulation is given down below, following sub−section will explain the conclusions which can be taken from this.

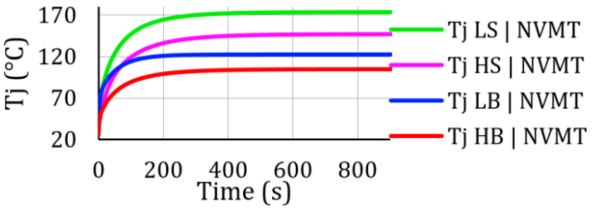


Figure 21. PCB Definition Simulation Results

Influence of Copper Thickness on PCB

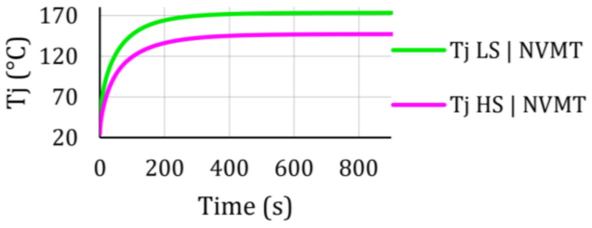


Figure 22. Copper Thickness Influence

The simulation shows that the higher Tj is found on the PCB with the lower copper thickness.

As explained before, a higher temperature for the same stress on the MOSFET shows a worse heat dissipation. Having more copper in the PCB implies having less of the other material, which are worse thermal conductors (~390 W/mK copper vs ~1 W/mK isolating material in PCB (polyepoxyde and glass fiber))

This conclusion can be applied on a wider range: to improve thermal dissipation, a change of materials for the substrate is a good way to start (if matches mechanical needs).

Influence of PCB Copper Spreading Area

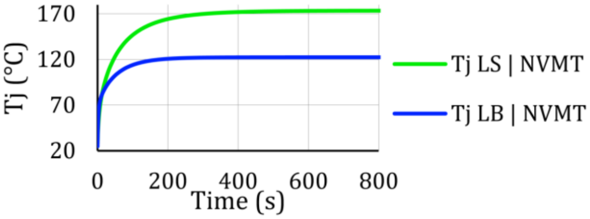


Figure 23. PCB Copper Area Influence

This curve shows the impact that a wider copper area has on the heat dissipation of the thermal energy. The wider copper area underneath the MOSFET, the lower the temperature.

This conclusion can also be used to build appropriate heatsinks for power MOSFETs. Indeed, the fans of a power heatsink are designed to ensure a maximum dissipation surface to match the thermal power needs of the component working.

MOSFET Characteristics and Housing Influence

From the previous section, we know that the bigger the PCB dissipation area, the lower the temperature. One could thus suppose that it is the same for MOSFETs: the wider the housing, the lower the temperature.

The following simulation is set to have around the same power through the MOSFETs (357W) and MOSFET housing directly linked to the ambient temperature (no Cauer network to model PCB, to ensure only the MOSFET is studied).

To achieve the wanted power, the current is distributed as followed:

212 amps in NVMT MOSFET
65 amps in NVMF MOSFET

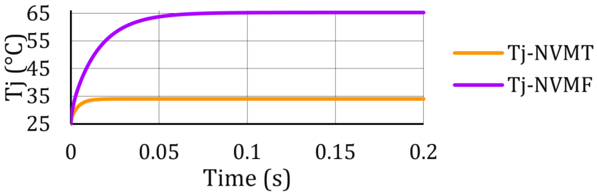


Figure 24. Junction Temperature Comparison between MOSFETs

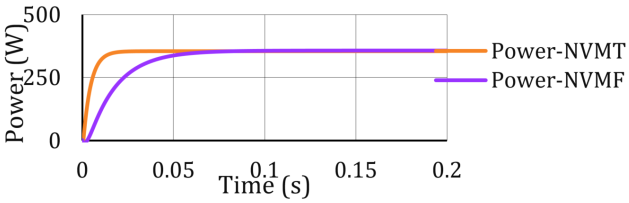


Figure 25. Power through MOSFETs

Although the same power is put through the MOSFETs, a 30°C difference in the stabilized temperature is seen.

This difference comes as expected from the difference of housing size, 8 x 8 mm for NVMT MOSFET, vs 5 x 6 mm for NVMF MOSFET. This result stresses out the fact that the important element for temperature is not the current through the MOSFET, but the power. See section "*More food for thoughts*" for more elements regarding the maximum current supported by MOSFETs vs the temperature.

Influence of Initial Temperature

The initial temperature of the application plays an important role into managing the temperature evolution of a component.

To study this, the simulation is run twice with the same set up except for initial temperature.

Full set–up description:

• Frequency	50 Hz
• Current	200 amps
• Duty Cycle	50%
• Trise = Tfall	1 ns
• Initial temperatures	140°C & −40°C
• MOSFET used	NVMT

Results and Analysis:

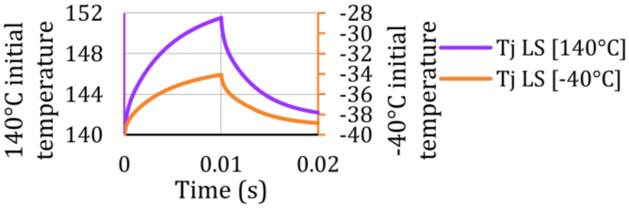


Figure 26. Initial Temperature Influence

Since MOSFET $R_{DS(ON)}$ increases with the temperature, the temperature increases with stress is even higher when the temperature at the start of the stress is higher.

In this example, the temperature increase is doubled between −40°C start and 140°C start:

$\Delta T_{-40^\circ C} = +6^\circ C$

$\Delta T_{140^\circ C} = +12^\circ C$

Therefore, worst case study for MOSFETs is at high temperature. In a complete application study however, other components need to be taken into account, with different thermal coefficient (main characteristic sensibility to temperature).

As a general conclusion, checking minimum and maximum initial temperatures behaviors of a complete application is the best way to ensure a full coverage of the thermal risk in the application.

And with SIMetrix it is really easy as just shown.

Influence of Stress Characteristics

Short Pulse

When studying the usual working mode of a MOSFET, time of study is usually far below the previously considered over 100 s. This section focuses on shorter stress time. What specifically characterizes a short pulse for thermal analysis will be discussed hereafter.

Full set−up description:

Frequency	20 Hz
Current	300 amps
Duty Cycle	50%
Trise = Tfall	1 ns
Initial temperatures	25°C
MOSFET used	NVMT

Results and Analysis:

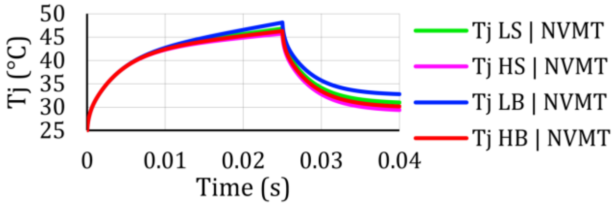


Figure 27. Short Pulse Junction Temperature Evolution

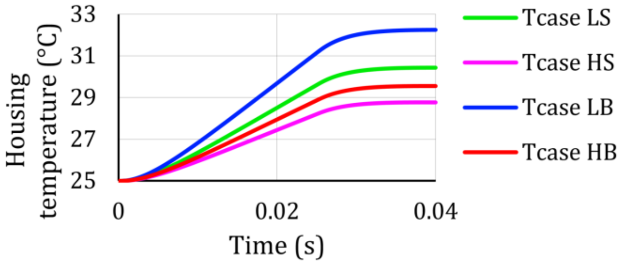


Figure 28. Short Pulse Housing Temperature Evolution

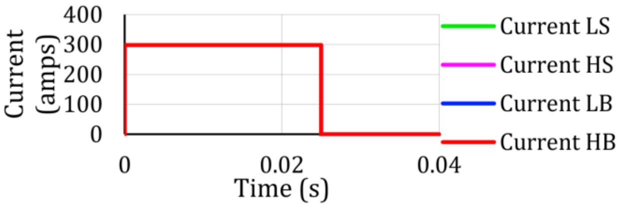


Figure 29. Short Pulse Current Temperature Evolution

In this simulation, the PCB variation does not impact temperature. This is what is considered as a “short pulse” thermal behavior wise. Indeed, the stress is so short that the heat transfer does not have enough time to spread through the different mechanical stacks of the assembly. As proof, under 10 ms, the temperature at the contact of the PCB has barely increased. As a result, in the different curves presented, under 10 ms, the same impact is seen on the junction temperature evolution.

All this means that to improve the thermal behavior of the MOSFET for short pulses, the focus must be put on MOSFETs. The only way to reduce temperature increase during short pulse is to reduce MOSFET $R_{DS(ON)}$ and contact resistances inside the housing.

Repetitive Stress

Full set−up description:

Frequency	0.3 Hz
Current	90 amps
Duty Cycle	20% to 100%
Trise = Tfall	1 ns
Initial temperatures	25°C
MOSFET used	NVMT

Results and Analysis:

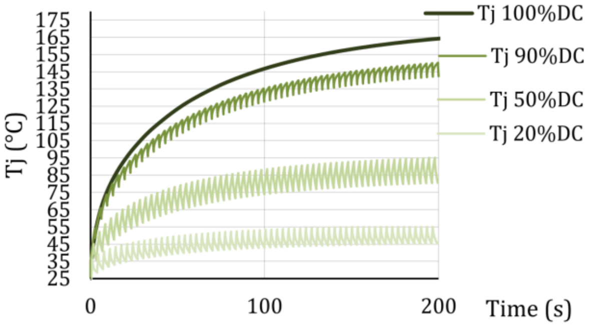


Figure 30. Repetitive Stress Junction Temperature Evolution

Temperature evolution for a MOSFET under a repetitive stress is a combination of the MOSFET behavior during short pulses and the general temperature evolution linked to substrate used, as long as the temperature at the end of one period is higher than the one at the beginning of the period.

For this example, we focused on the impact of the duty cycle used. A higher duty cycle means the MOSFET is stressed longer with less time to cool down. Therefore for the same current, the temperature evolution from 20% to 100% duty cycle cannot be neglected.

Linear Mode Command

Study duration in this section will be under 20 ms. The PCB is therefore ignored.

Full set−up description:

• Current	200 amps
• Trise = Tfall (fast command)	100 ns
• Trise = Tfall (linear command)	10 µs
• Initial temperatures	25°C
• MOSFET used	NVMT

Results and Analysis:

Slow commutation may be required for MOSFET in some applications, often to avoid electro−magnetic pulse issues. However, a slow MOS opening can lead to MOS driven in linear mode. In linear mode, the MOSFET resistance is not the typical $R_{DS(ON)}$ given in the datasheet, but a changing one depending on the gate voltage. For more information on on−region studies, see papers in reference [6] and [7]

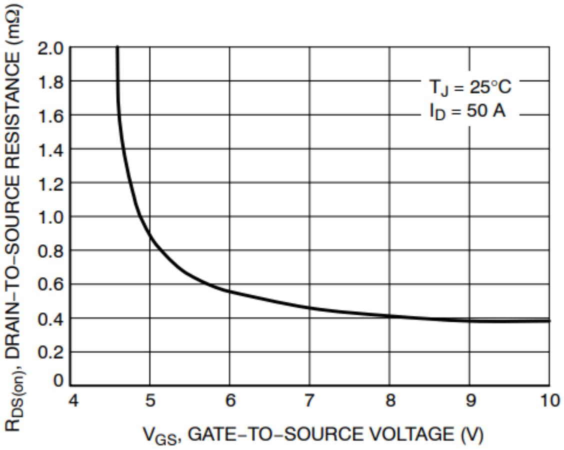


Figure 31. $R_{DS(ON)}$ vs. V_{GS} Curve from NVMT MOS Datasheet

This resistance being higher than the $R_{DS(ON)}$, a special attention needs to be put regarding increased temperature while the MOSFET is switching.

Throughout a slow command, the resistance will slowly decrease until it reaches the $R_{DS(ON)}$.

A simulation is run to see the precise impact of the linear mode on the MOSFET temperature.

Single pulse:

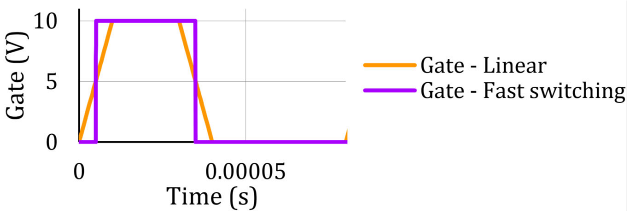


Figure 32. Gate Voltage − Linear vs. Fast Switching

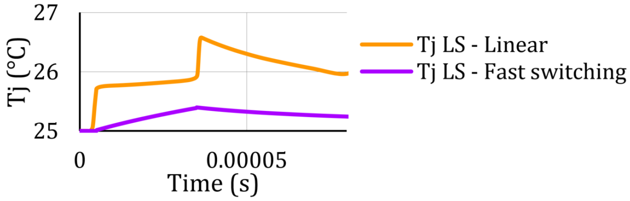


Figure 33. Junction Temperature − Linear vs. Fast Switching

As shown in the simulation results, the temperature of a MOSFET driven slowly increases faster than the one of a fast switched MOSFET.

The evolution of the temperature corresponds to the $R_{DS(ON)}$ evolution: as the gate voltage goes over the MOSFET threshold voltage, the MOSFET channel opens with the high resistance presented in Figure 31. Thus the temperature rises along a step. As the gate voltage goes higher, the resistance decreases and therefore the power going through the MOSFET decreases. The temperature still increases since the power is still flowing through the MOSFET, except five times smaller. As the MOSFET closes, the same evolution is seen, thus another step in the temperature.

For a single impulse this is not really an issue as long as the temperature increase keeps the MOSFET under the allowed range for the MOSFET.

Regular application (repetitive command):

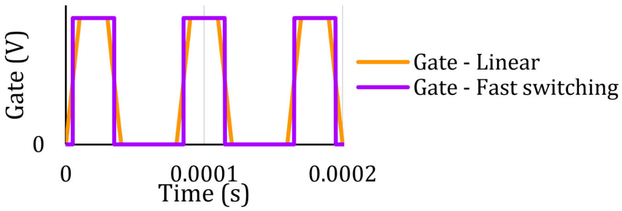


Figure 34. Gate Voltage − Linear vs. Fast Switching (zoomed)

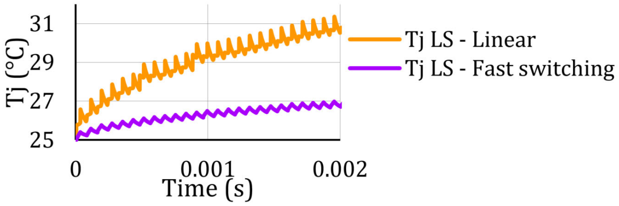


Figure 35. Junction Temperature − Linear vs. Fast Switching

In the case of a regular application, where the MOSFET is switched on and off in cycles, the small additional temperature steps from linear command add up faster than the PCB can cool down the MOSFET. The increase of temperature for a linear switching is significantly higher than the one for a fast switch. The curve in Figure 36 shows the difference of general slope difference for these increase on the long run.

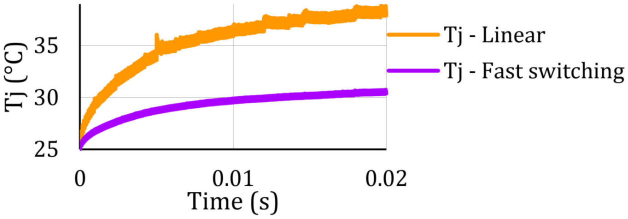


Figure 36. Junction Temperature − Linear vs. Fast Switching

MORE FOOD FOR THOUGHTS

In this application note, the possibilities of SIMetrix to study thermal behaviors are explored for the main situations of MOSFET command.

However, when building a justification folder for an application, a comparison with datasheet is always required. It is still important to keep a critical view on both datasheet specification conditions as well as not solely rely on simulation for the component validation. Indeed, we will see in this section that the junction temperature is not always the critical value for a MOSFET application limits. We will also see that datasheet test conditions are not to be neglected when considering an application working at the maximum rated values of the component.

Maximum Pulse Drain Current:

As a first example, down below the datasheet extract with the max pulsed drain current:

Table 5. PULSED DRAIN CHARACTERISTICS FOR NVMT MOSFET

Pulsed Drain Current	T_A = 25°C, t_p = 10 μs	I_{DM}	900	A

A simulation is run for this set up:

Current	900 amps
Initial temperatures	25°C
MOSFET used	NVMT
PCB used	LS

The results are as follow:

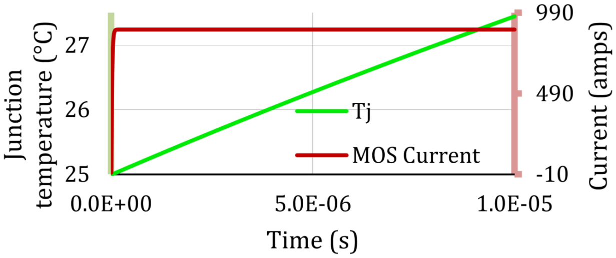


Figure 37. Temperature and Current with Max Drain Current Input

Junction temperature of the MOSFET does not rise high enough to raise any concern for the MOSFET integrity. The simulation is ran inside the datasheet specification so the model is accurate. This means the maximum current limitation does not directly come from the MOSFET $R_{DS(ON)}$, but from other MOSFET characteristics, or rather from the assembly itself. MOSFETs are assembled in their resin housing using power connectors. These connectors are made of copper or aluminum, with their own resistance, Joule losses and temperature increase with joule losses. A really high current usually overstresses these metals before it overstresses the MOSFET die. The temperature can bring damage to the connectors, the MOSFET or the housing itself, with no specific increase of the MOSFET junction temperature.

Once again, this shows that a thermal model stays valid only inside the datasheet limits, regardless of the apparent well−being of the die from a simulation point of view. Extrapolation should not be considered from a simple simulation result.

Continuous Drain Current:

As a second example, here is the maximum continuous current supported by the MOSFET:

Table 6. MAXIMUM CONTINUOUS DRAIN CURRENT @ TC = 25°C FOR NVMT MOSFET

Continuous Drain Current $R_{\theta JC}$	T_C = 25°C	I_D	558	A

A new simulation is run with the same conditions as specified here:

Current	558 amps
Initial temperatures	25°C
MOSFET used	NVMT
PCB used	LS

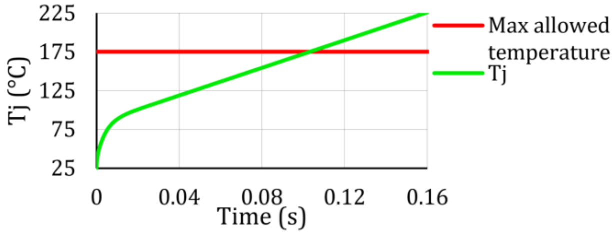


Figure 38. Temperature with Continuous Drain Current @ Stable T_C

The MOSFET temperature increases in only a few milliseconds above the maximum allowed temperature for this MOSFET.

The error which was willingly made here was to simulate with the following hypothesis: the Tc temperature in the datasheet is the ambient temperature for our complete set up with PCB. It is in fact not the case: the temperature given in the datasheet is the housing temperature (Tc). Now if the PCB is removed and the MOSFET housing is directly connected to a 25 V source representing the 25°C, the simulation gives the following result:

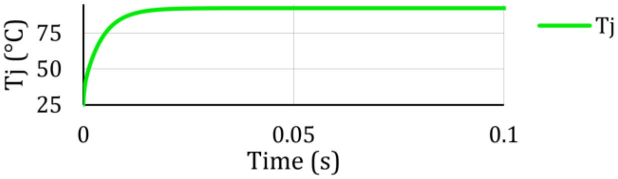


Figure 39. Temperature with Continuous Drain Current @ Stable T_A

The temperature raises up to over 90°C with what can be called a perfect heatsink: the voltage source ensures the temperature of the MOSFET housing is kept at 25°C. In a real life application, it is almost impossible to find a cooling solution efficient enough to achieve the same results.

For a more relevant maximum current for applications, users can use the continuous drain current given for the ambient temperature:

Table 7. MAXIMUM CONTINUOUS DRAIN CURRENT @ T_A = 25°C FOR NVMT MOSFET

Continuous Drain Current $R_{\theta JA}$	T_A = 25°C	I_D	79.8	A

CONCLUSION

SIMetrix is a powerful tool to study thermal behaviors in electronics. If a Cauer model is provided to simulate the thermal characteristics of mechanical stack−ups, complete thermal studies can be done.

It is a versatile tool which can be used to simulate full life cycles, check MOSFETs characteristics, and optimize the working range of the application.

Through these studies, the impact of the dissipation area (the wider the best) is easily shown.

The heat transfer speed can also be studied, to see the impact of short pulses, for which wide dissipation areas are not relevant, since the heat transfer does not reach the dissipation point.

The thermal simulation also helps to understand maximum stresses allowed on datasheet. Improving dissipation is not the go−to solution for all specification mismatch: some are driven by other components in the MOSFET housing assembly, some can simply not be achieved efficiently.

REFERENCES

MOSFET models:

[1] “A Physically Based Scalable SPICE Model for Shielded Gate Trench Power MOSFETs”

[2] “A Physically Based Scalable SPICE Model for Silicon Carbide Power MOSFETs”

[3] “Physically Based, Scalable SPICE Modeling Methodologies for Modern Power Electronic Devices”

SIMetrix:

[4] “SPICE Modeling Tutorial”

[5] “How to use Physical and Scalable Models with SIMetrix, OrCAD and LTSpice”

[6] “Using Physical and Scalable Simulation Models to Evaluate Parameters and Application Results”

[7] “Simulate with Physical and Scalable Discrete Models...What could we get ?”

In 2020, she joined **onsemi** as field application engineer.

ANNEX

Script to run several simulations at the same time using one schematics

```
*(green: comment) Set up to keep curves and Cauer networks up
during the simulation
NoUndo
*Read the Cauer network values file
Let lines = ReadFile('heatsinkData.txt')
*Variable initialisation
Let numLines=Length(lines)
Let refs=''
Let refIdx=0
*check the names of all the resistance and capacitances
for idx=0 to numLines-1
        if lines [idx]<>'' then
*in data file: line 1 is Resistance etc
                Let firstDataLine=idx+1
                Let refs=Parse(lines[idx])
                exit for
        endif
next idx
Let numRefs=Length(refs)
*run simulations for each Cauer network
for idx=firstDataLine to numLines-1
        Let num=idx-1
        if lines[idx]<>'' then
                Let values=Parse(lines[idx])
                for refidx=0 to numrefs-1
                        Unselect
                        Select /prop REF{refs[refidx]}
                        Prop VALUE {values[refidx]}
                next refidx
                Netlist netlist1.net
                Run netlist1.net
        endif
next idx
*Remove all done operation (Cauer networks modifications)
Undo
```

Cauer Models for Simulations

NVMTS0D4N04CL				NVMFS5C456NL			
HB				HB			
Copper Area	800	[mm^2]		Copper Area	800	[mm^2]	
Copper Thickness	2	[oz/ft^2]		Copper Thickness	2	[oz/ft^2]	
R		C		R		C	
R1	3.272964	C1	0.147195	R1	3.028952	C1	0.026106
R2	5.110304	C2	0.261917	R2	5.044262	C2	0.041564
R3	10.61216	C3	2.320184	R3	6.26684	C3	0.683576
R4	2.598421	C4	3.715802	R4	5.856597	C4	0.963063
R5	3.714489	C5	7.002144	R5	0.521666	C5	7.894196
R6	0.950235	C6	94.94824	R6	7.043194	C6	1.031262
LB				LB			
Copper Area	800	[mm^2]		Copper Area	800	[mm^2]	
Copper Thickness	1	[oz/ft^2]		Copper Thickness	1	[oz/ft^2]	
R		C		R		C	
R1	7.554073	C1	0.090167	R1	3.034604	C1	0.026084
R2	9.258183	C2	0.093982	R2	5.082504	C2	0.041402
R3	6.464182	C3	2.271137	R3	8.674736	C3	0.637441
R4	5.959158	C4	2.120965	R4	6.510921	C4	0.702989
R5	2.010848	C5	13.84377	R5	1.045653	C5	5.227638
R6	0.062748	C6	1872.721	R6	10.13882	C6	0.512838
LS				LS			
Copper Area	350	[mm^2]		Copper Area	350	[mm^2]	
Copper Thickness	1	[oz/ft^2]		Copper Thickness	1	[oz/ft^2]	
R		C		R		C	
R1	4.653994	C1	0.122885	R1	3.18942	C1	0.025438
R2	10.35295	C2	0.146819	R2	5.729411	C2	0.037962
R3	9.840158	C3	0.676801	R3	13.33134	C3	0.380225
R4	12.6698	C4	1.323351	R4	9.146856	C4	0.47707
R5	5.951378	C5	6.285148	R5	1.274545	C5	5.090209
R6	1.146383	C6	86.42503	R6	11.30379	C6	0.418588
HS				HS			
Copper Area	350	[mm^2]		Copper Area	350	[mm^2]	
Copper Thickness	2	[oz/ft^2]		Copper Thickness	2	[oz/ft^2]	
R		C		R		C	
R1	2.357675	C1	0.179096	R1	3.184889	C1	0.025452
R2	6.361766	C2	0.402543	R2	5.684675	C2	0.038089
R3	15.6095	C3	0.717652	R3	9.909104	C3	0.403145
R4	4.942365	C4	1.735005	R4	8.636391	C4	0.625307
R5	7.24529	C5	5.436704	R5	0.691183	C5	7.219813
R6	1.48013	C6	59.40064	R6	8.245533	C6	0.769009

onsemi, onsemi, and other names, marks, and brands are registered and/or common law trademarks of Semiconductor Components Industries, LLC dba "**onsemi**" or its affiliates and/or subsidiaries in the United States and/or other countries. **onsemi** owns the rights to a number of patents, trademarks, copyrights, trade secrets, and other intellectual property. A listing of **onsemi**'s product/patent coverage may be accessed at www.onsemi.com/site/pdf/Patent−Marking.pdf. **onsemi** reserves the right to make changes at any time to any products or information herein, without notice. The information herein is provided "as−is" and **onsemi** makes no warranty, representation or guarantee regarding the accuracy of the information, product features, availability, functionality, or suitability of its products for any particular purpose, nor does **onsemi** 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 **onsemi** products, including compliance with all laws, regulations and safety requirements or standards, regardless of any support or applications information provided by **onsemi**. "Typical" parameters which may be provided in **onsemi** 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. **onsemi** does not convey any license under any of its intellectual property rights nor the rights of others. **onsemi** products are not designed, 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 **onsemi** products for any such unintended or unauthorized application, Buyer shall indemnify and hold **onsemi** and its officers, employees, subsidiaries, affiliates, and distributors harmless against all claims, costs, damages, and expenses, and reasonable attorney fees arising out of, directly or indirectly, any claim of personal injury or death associated with such unintended or unauthorized use, even if such claim alleges that **onsemi** was negligent regarding the design or manufacture of the part. **onsemi** is an Equal Opportunity/Affirmative Action Employer. This literature is subject to all applicable copyright laws and is not for resale in any manner.

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Email Requests to: orderlit@onsemi.com

onsemi Website: www.onsemi.com

TECHNICAL SUPPORT
North American Technical Support:
Voice Mail: 1 800−282−9855 Toll Free USA/Canada
Phone: 011 421 33 790 2910

Europe, Middle East and Africa Technical Support:
Phone: 00421 33 790 2910
For additional information, please contact your local Sales Representative

◊