



PLECS

Tutorial

Introduction to the Thermal Domain in PLECS

Tutorial Version 1.0

www.plexim.com

- ▶ Request a PLECS trial license
- ▶ Check the PLECS documentation

1 Introduction

In this exercise you will learn how to use the thermal modeling features of PLECS to create a combined electrical-thermal simulation of a buck converter. The specific learning goals for this exercise are listed as follows:

- Learn which components are needed for a thermal simulation.
- Create a simple thermal model for the MOSFET with conduction and switching losses. A more advanced procedure with using the PLECS import wizard is described in the tutorial “104 - PLECS: Thermal Simulation of a Buck-Converter”.
- Establish the steady-state operating junction temperature of the MOSFET.
- Calculate the individual and total losses of the switches and overall system efficiency.

Before you begin Ensure the file `thermal_domain_start.plecs` and the directory `thermal_lib` are all located in your working directory. You should also have the reference files that you can compare with your own models at each stage of the exercise. A datasheet `c3m0120090d.pdf` is also included for your reference as part of the exercise.

2 Circuit

The circuit you will use for this exercise is shown in Fig 1. It is a buck converter with a SiC MOSFET switch that operates at 20 kHz. The electrical parameters are given in Table 1.

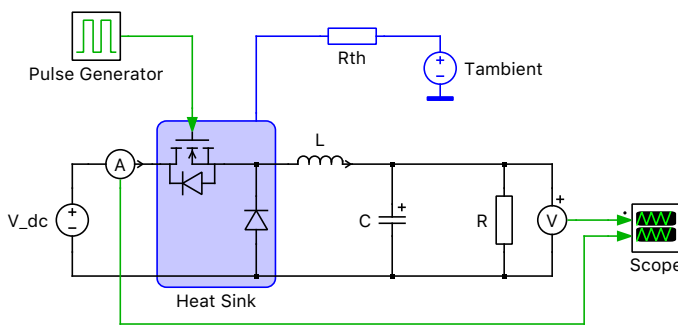


Figure 1: Combined electrical-thermal simulation of a buck converter

3 Background

To perform a thermal simulation using PLECS, three components are needed: a Heat Sink, semiconductor component(s) with a thermal description and an Ambient Temperature. The thermal simulation takes place at the same time as the circuit simulation but the thermal components are represented in PLECS in a separate domain. The thermal quantities and their analogous electrical equivalents are given in Table 2. In the analogous electrical circuit, conduction losses of a component are modeled as an ideal current source, the thermal impedance between the component and air as a simple resistor or RC chain, and the ambient temperature as a voltage source. This equivalent circuit is shown in Fig. 2.

Table 1: Buck converter electrical parameters

Component	V_{DC}	f_{sw}	P_{out}	Duty cycle	C	$V_{C,initial}$	L	$I_{L,initial}$	R
Value	500 V	20 kHz	2 kW	50 %	200 μ F	250 V	10 mH	8 A	31.25 Ω

The Heat Sink block is the fundamental component for modeling thermal circuits in PLECS. It provides a surface of equal temperature that can represent an actual heat sink in a system but more generally is an individual thermal layer. The heat sink block absorbs the switching and conduction losses of all components contained within its boundary. In PLECS, the switching energy pulses are modeled as a dirac-type pulses, having zero width and an infinite height. Therefore, either the thermal capacitance of the heat sink must be defined or a thermal chain with capacitance must be used to prevent the switching energy pulses from creating an infinite temperature across the thermal resistance.



Note: The Heat Sink captures *all* intrinsic components that dissipate thermal losses, meaning that in addition to semiconductor losses, ohmic resistors can also be monitored. Ohmic losses are calculated as $i^2 \cdot R$, or V^2/R .

Table 2: Equivalent thermal and electrical quantities

Therm. Domain	Elec. Domain
Temperature (K)	Voltage (V)
Heat (J)	Charge (C)
Heat flow (W)	Current (A)
Therm. cap. (J/K)	Elec. cap. (F)
Therm. res. (K/W)	Elec. res. (Ω)

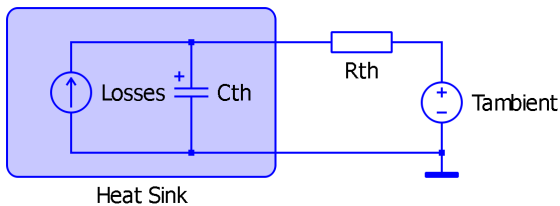


Figure 2: Electrical equivalent of thermal circuit

4 Tutorial

4.1 Create the thermal model



Your Task: Open the PLECS model `thermal_domain_start.plecs` and add a Heat Sink, Thermal Resistor and Constant Temperature (Grounded) block to the circuit, as shown in Fig. 1. These components can be found in the Thermal section of the PLECS component library. The parameters for these components are as follows:

- The ambient temperature is 25 °C.
- The thermal capacitance of the heat sink is 0.001 J/K. Also set the initial temperature to the same value as the ambient temperature.
- The thermal resistance between the heat sink and the ambient temperature is unclear in the current design state. It is a common strategy to first set the thermal resistance to the maximum allowable value from an electrical design perspective. Assuming that the efficiency is at least 95 %, this results in a maximum power dissipation of 100 W. With a maximum temperature change of 80 °C, this leads to an initial estimation of 0.8 K/W.



Note: You can move a heat sink terminal to another free slot on the border by dragging it while holding down the Shift key. When you release the mouse button, the terminal is moved.

4.2 Link and create a thermal description

The losses of the diode have been predefined in the file C3D08060A.xml. To add this thermal description to the diode, you must add the path of this file to the thermal description search path. The MOSFET losses are created from scratch with some input values. After completing the loss description, use the PLECS probe to monitor the signals associated with the MOSFET and the diode.



Your Task: In this task you will load the predefined thermal description for the diode.

- 1 Under the **File + PLECS Preferences...** menu, navigate to the **Thermal** tab and add the directory that contains the thermal description to the search path. Press the **Rescan** button to activate the change.
- 2 Double-click the Diode component. Under the **Thermal description** parameter on the **Thermal** tab, select “C3D08060A” using the **From library...** option. Click **Apply** to save the selection.



Your Task: In this task you will create a new thermal description for the MOSFET with Diode using a few values from the data sheet.

- 1 Double-click the MOSFET with Diode component. Under the **Thermal description** parameter on the **Thermal** tab, select **New Thermal description...**
- 2 Add “Wolfspeed” as manufacturer and “C3M0120090D” as part number. Save the file as C3M0120090D_tutorial.xml in the local thermal library that you defined in the previous task.
- 3 Navigate to the tab **Turn-on loss** and set the **Energy scale** to μJ .
- 4 The next step is to fill the table with meaningful loss data, which can be found in the data sheet in **Figure 23** and **24**. These loss data are measured with a constant gate resistor R_g and a constant gate-source voltage, while V_{DD} changes from 600 V to 400 V. Since the buck converter switches 500 V, it makes sense to include both values in the thermal description for better interpolation. Also, loss values for different currents should be chosen. Since the target current is 8 A, this value and 15 A are chosen.
- 5 To ensure good interpolation, it is recommended to add negative current and voltage values with the switching energy of $0 \mu\text{J}$.
- 6 Right-click on the x and y axis and add **New current values...** and **New voltage values...** respectively. Fill in the following numbers:

	-10 A	0 A	8 A	15 A
-100 V	$0 \mu\text{J}$	$0 \mu\text{J}$	$0 \mu\text{J}$	$0 \mu\text{J}$
0 V	$0 \mu\text{J}$	$0 \mu\text{J}$	$0 \mu\text{J}$	$0 \mu\text{J}$
400 V	$0 \mu\text{J}$	$0 \mu\text{J}$	$70 \mu\text{J}$	$140 \mu\text{J}$
600 V	$0 \mu\text{J}$	$0 \mu\text{J}$	$125 \mu\text{J}$	$220 \mu\text{J}$

- 7 Navigate to the tab **Turn-off loss** and repeat all mentioned points.

- 8** So far, only loss data for 25 °C has been added. The addition of another temperature value is essential for a temperature dependent simulation. **Figure 26** in the datasheet shows temperature-dependent switching losses. For simplicity, one could assume a linear behavior and compare the losses at 25 °C and 150 °C. Turn-off losses increase by only 20 %, while turn-on losses increase by about 80 %. For the sake of convenience we simply multiply the values with these two factors, knowing that it is not entirely correct. Do not forget to add the zero entries as well.

		25 °C		150 °C	
		8 A	15 A	8 A	15 A
Turn-on loss	400 V	70 μ J	140 μ J	126 μ J	252 μ J
	600 V	125 μ J	220 μ J	225 μ J	396 μ J
Turn-off loss	400 V	8 μ J	20 μ J	9.6 μ J	24 μ J
	600 V	15 μ J	35 μ J	18 μ J	42 μ J

- 9** Using **Figure 2** and **3** of the device’s datasheet it is possible to estimate the conduction losses. It is recommended by the manufacturer to refer to the “ $V_{GS} = 15$ ” curve unless you are not driving the gate with full voltage.
- 10** Navigate to the tab **Conduction loss** and input the values. Ensure there is a value of 0 V at 0 A for all temperatures. Fill in the numbers of the following table:

	0 A	5 A	15 A	25 A	35 A	45 A
25 °C	0 V	0.5 V	1.8 V	3.3 V	5 V	7.1 V
150 °C	0 V	0.8 V	2.6 V	4.8 V	7.4 V	10.7 V

- 11** In addition, the conduction loss for the 3rd quadrant must be defined. Even though the buck converter in this configuration is not capable of conducting in the negative current region, it is good practice to at least define the body diode conduction. For an application where conduction is important for negative currents, such as a synchronous buck converter, it is recommended to define the gate dependent conduction losses (Appendix 8).

Figures 9 and **10** provide data for the body diode at 25 °C and 150 °C, respectively. Since a SiC MOSFET is normally turned off with negative gate voltage, the curves for $V_{GS} = -4$ V are taken and added to the matrix:

	-40 A	-30 A	-20 A	-10 A	-5 A	-1 A	-0.1 A
25 °C	-7.6 V	-6.8 V	-6 V	-5 V	-4.3 V	-3.4 V	-2.8 V
150 °C	-6.7 V	-6.1 V	-5.1 V	-4.5 V	-3.8 V	-3 V	-2.4 V

- 12** Navigate to the **Thermal Chain** tab, select “Cauer” as the type, and increase the number of elements to 1. Enter 0.5 K/W and 0.1 J/K as R and C , respectively.

4.3 Thermal Simulation



Your Task: Now we can monitor the MOSFET junction temperature and run a simulation.

- 1 Close the Thermal Editor window and return to the schematic. Add a Probe block from the library to your schematic and use it to monitor the MOSFET temperature. The easiest way to associate a component with a Probe is to drag it directly on to the Probe and release the mouse button when the pointer shape changes to crosshairs. After doing this, select the option to monitor the junction temperature from the list on the right.
- 2 Enter a value of 25 °C for the **Initial temperature** on that same tab for the MOSFET, and also for the diode.
- 3 Run the simulation for 1.0 s by pressing **Ctrl + T**, leaving all other simulation parameters as their defaults.



What is the final temperature of the MOSFET junction?



~ 37.6 °C



At this stage, your model should be the same as the reference model: thermal_domain_1.plecs.

4.4 Average Loss Calculation

In many cases, a factor of interest is the average power dissipation of each semiconductor. The average losses for a component can be calculated by summing the losses that occur during a switching cycle and generating an average power pulse during the next switching cycle. This cycle-average loss calculation procedure is summarized in Fig. 3.

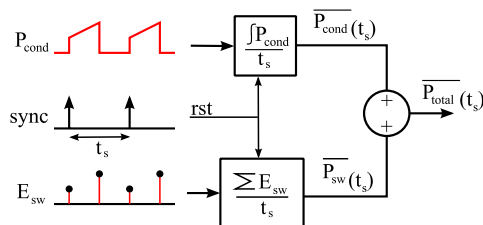


Figure 3: Calculation of total cycle-average losses




Your Task:

- 1 The PLECS component library contains a block in the “System” list that can be used to measure all losses. The Switch Loss Calculator is suited for determining average conduction and switching losses of semiconductors. Place two Switch Loss Calculator components into the model and provide the appropriate loss signals for the MOSFET and diode. To do this drag each of the semiconductors into the corresponding loss calculator.
- 2 The Switch Loss Calculator is an intrinsic component. To see its internal implementation, right-click the component and select **Help**.
- 3 In the Switch Loss Calculator set the **Averaging time** to 50e-6 s which is the inverse of the switching frequency of 20 kHz.

- 4 From the Output signals select Total loss to get the averaged total loss for the semiconductor.
- 5 Place a Heat Flow Meter block between the thermal chain and the ambient temperature to measure the total average losses of both semiconductors. Use a Signal Multiplexer block to display the total losses on the same scope used to display the calculated losses.
- 6 Run the simulation for 1.0 s. Compare the output of the heat flow meter with the total loss calculation.

? Do the results agree? Why is there a difference in the wave forms?

A ~12 W. There is a difference because the heat flow meter measures the heat flow in the thermal circuit, where it takes several seconds for the heat sink to warm up due to its capacitance. The loss probes, however, are connected directly to the components' junctions.

 At this stage, your model should be the same as the reference model: thermal_domain_2.plecs.

5 Efficiency Calculation

Once the losses associated with the power semiconductor devices are known, the efficiency of the converter can be calculated. In this tutorial only power semiconductor devices are placed on top of the Heat Sink component. However, also electrical resistors can be placed on a heatsink to take into account ohmic losses.

Your Task:

- 1 Place an additional Probe, Periodic Average, and Multiplexer block into the schematic near where the total MOSFET and diode losses are summed. Probe the input power of the converter by monitoring the power signal of the DC Voltage Source. The **Averaging time** of this block should also be 0.05 ms. Using the Multiplexer, combine the source power measurement and total loss calculation of the switching cell into a single bus with a width of 2 signals, as shown in Figure 4. As an optional step, add a Saturation block with a **Lower limit** of 1 and an **Upper limit** of inf, which ensures that there are no “Division by zero” issues at the beginning of the simulation.
- 2 Place a Function block in series with the output of the Multiplexer and Saturation blocks. Here you will use the vectorized bus to access both the source power and switching cell loss signals. In the **Expression** box, type the equation $(1 - u[1] / u[2]) * 100$. Assuming that the upper (first) signal you routed to the Multiplexer input is the source power, this efficiency equation is equivalent to

$$\eta = (P_{in} - P_{loss}) / P_{in} .$$

Multiplying this result by 100 effectively formats the result as a percentage. Finally, a Display block can be used to show the calculation result in numerical form, which may be preferred over using a Scope to show the calculation as a simulated waveform (and this may also be desired for the previous calculation for the total combined power of the MOSFET and diode losses).

- 3 Rerun the simulation.

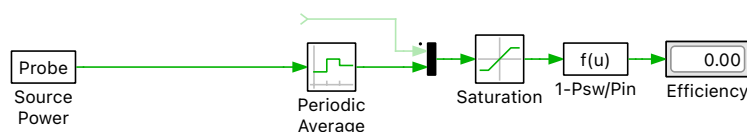


Figure 4: Calculating the converter efficiency using the input power and total device losses



Note: If you specify the Thermal description parameter for a semiconductor component, the dissipated thermal power does not correspond to the electrical power that is consumed by the device. Also, due to the instantaneous nature of the switching transitions, the dissipated thermal energy cannot be consumed electrically by the device. These important factors must be taken into account when you use the thermal losses for estimating the efficiency of a circuit. This means that all on-resistance and forward voltage values specified by a user in the component parameters are fixed during a simulation and only influence the electrical behavior of the system, i.e., the voltage and current magnitudes. Therefore, the efficiency calculation technique described above is the correct method you need to use in PLECS, and simply calculating $P_{\text{out}}/P_{\text{in}}$ is not equivalent as it doesn't account for the thermal losses in the system.



Your Task: (Optional) You can add on-resistance and forward voltage to the semiconductor devices to account for the voltage drop across these components.

1 Place an additional Probe, Periodic Average, and Multiplexer block into the schematic near where the total MOSFET and diode losses are summed. Probe the voltage across the diode and the current through the inductor. Note that the voltage across the diode will be negative, therefore, add a gain with the value -1 . Multiply both signals and average the results with 0.05 ms. Using the Multiplexer, combine the source power and output power measurement into a single bus with a width of 2 signals, as shown in Figure 5. As an optional step, add a Saturation block with a **Lower limit** of 1 and an **Upper limit** of inf , which ensures that there are no “Division by zero” issues at the beginning of the simulation.

2 Place a Function block in series with the output of the Multiplexer and Saturation blocks. Here you will use the vectorized bus to access both the source and output power signals. In the **Expression** box, type the equation $(u[1]/u[2])*100$. Assuming that the upper (first) signal you routed to the Multiplexer input is the source power, this efficiency equation is equivalent to

$$\eta = P_{\text{out}}/P_{\text{in}}.$$

Multiplying this result by 100 gives the result as a percentage. Finally, a Display block can be used to show the efficiency as a number, which might be more convenient than a waveform in a Scope.

3 Rerun the simulation. You will notice that the result is 100.00 % since no electrical losses are modeled.

4 The on-resistance of the MOSFET can be evaluated with the values provided in the conduction loss table. If the temperature is assumed to be fix at 25 °C and knowing that the average current through the semiconductor is 4 A, the resistance can be calculated at the given point at 5 A as

$$R_{\text{on}} = 0.5 \text{ V}/5 \text{ A} = 0.1 \Omega.$$

5 Similarly the diode's forward voltage and on-resistance can be estimated. The forward voltage can be found in the curve where the diode starts to conduct. For the 25 °C curve this is at 0.24 A with a forward voltage is 0.8877 V. The on-resistance can be found as the slope of the straight line as

$$R_{\text{on}} = (2.341 \text{ V} - 0.9977 \text{ V})/(19.9 \text{ A} - 0.24 \text{ A}) = 0.0739 \Omega.$$



What is the difference compared to the efficiency calculated whit thermal losses?



Since the converter is already very efficient ($> 99\%$) the additional electrical parameter do not influence the simulation result significantly.



At this stage, your model should be the same as the reference model: thermal_domain_3.plecs.

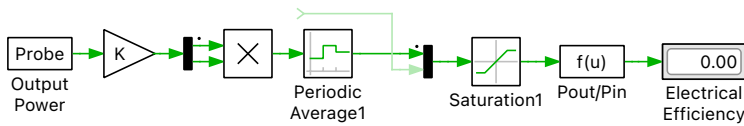


Figure 5: Calculating the electrical efficiency using the input power and output power



Your Task: (Optional) You can load the thermal description from Wolfspeed and compare to the generated simpler model.

- 1 For later comparison, first save the simulation results in the Sscope.
- 2 Double-click the MOSFET and load the thermal description C3M0120090D.xml.
- 3 This topic is not covered in this introductory tutorial, but a thermal description can also use custom variables, such as for allowing gate resistance-dependence in switching loss data. The MOSFET data already includes a variable “Rg” in the equations for **Turn-on losses** and **Turn-off losses**. In order to run a simulation we need to provide a value for the gate resistance. Double-click the MOSFET component. Under the **Thermal description** parameter on the **Thermal** tab, enter a value of 2.5Ω for the **Gate resistance**.
- 4 Enter a value of 25°C for the **Initial temperature** on that same tab for the MOSFET, and also for the diode.
- 5 Run the simulation for 1.0 s by pressing **Ctrl + T**, leaving all other simulation parameters as their defaults.

6 Steady-state operation

For an efficiency and loss estimation, the steady-state of a circuit is of primary interest. Since larger systems require a lot of simulation power until they reach a steady-state after the initial transients, it is better to perform a steady-state analysis directly.

From the **Simulation** menu choose **Analysis tools...** This opens a dialog, in which the analysis has already been pre-configured. To start the analysis, click on the **Start analysis** button. You can display the progress of the analysis by clicking on the **Show log** button.

When the analysis has finished a simulation of three steady-state cycles is displayed for all wave forms pre-configured in the Scope.

? What is the efficiency of the system when operating at steady state?

A For both the simple thermal description and the complete thermal description the calculated efficiency is 99.42%. This should emphasize the fact that even when quickly setting up a thermal description with only a few data points one can already obtain meaningful simulation results.



At this stage, your model should be the same as the reference model: thermal_domain_4.plecs.

7 Conclusion

In this exercise, you learned how to form a combined electrical-thermal simulation using PLECS. The fundamental component required for thermal analysis is the heat sink. You also learned how to create

a thermal description for a switch. Once you have data describing the losses of the semiconductors and their thermal coupling in the thermal circuit model, you can then calculate system losses and efficiency.

An extension to this exercise is to practice designing more complex, hierarchical thermal models in PLECS. This can be achieved by layering thermal components using subsystems and defining heat sinks underneath the top mask to represent things such as the heat sink(s), hot spots, etc.

8 Appendix: Gate dependent conduction losses

A MOSFET can conduct current in both directions, and so the conduction losses of a MOSFET with integrated anti-parallel diode depend on the gate signal when the current flows in reverse direction. To account for this effect, you can provide two separate conduction loss definitions, one of which is used when the gate signal is non-zero, and the other, when the gate signal is zero.

To create separate tabs that define the conduction losses with respect to the gate signal, select the **MOSFET with Diode** device type, right-click on the tab bar and select **Use gate dependent conduction losses** from the context menu as depicted in Figure 6.

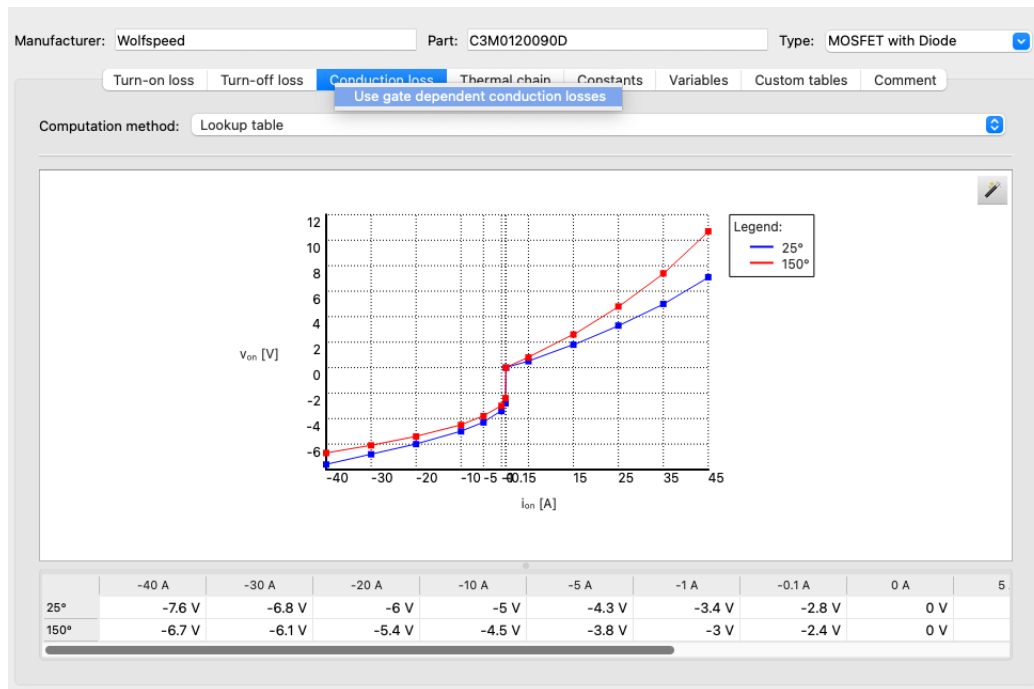


Figure 6: Setting up gate dependent conduction losses for a MOSFET with Diode

The data sheet for the C3M0120090D device contains the required data for the body diode used for the data set **Cond. loss (gate off)** in **Figures 8 - 10**. **Figures 13 - 15** provide the data for the 3rd quadrant characteristics used for **Cond. loss (gate on)**. If the latter data is missing in a data sheet, it is advisable to mirror the data for the MOSFET channel and parallelize it with the body diode data.

Revision History:

Tutorial Version 1.0 First release

How to Contact Plexim:

☎	+41 44 533 51 00	Phone
	+41 44 533 51 01	Fax
✉	Plexim GmbH Technoparkstrasse 1 8005 Zurich Switzerland	Mail
@	info@plexim.com	Email
	http://www.plexim.com	Web

PLECS Tutorial

© 2002–2022 by Plexim GmbH

The software PLECS described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from Plexim GmbH.

PLECS is a registered trademark of Plexim GmbH. MATLAB, Simulink and Simulink Coder are registered trademarks of The MathWorks, Inc. Other product or brand names are trademarks or registered trademarks of their respective holders.