Presented at the COMSOL Conference 2009 Milan

October 14-16 2009, Milan - Italy

COMSOL CONFERENCE 2009 MILAN

THERMAL DESIGN OF POWER ELECTRONIC DEVICES AND MODULES

N. Delmonte, M. Bernardoni, P. Cova and R. Menozzi

Dipartimento di Ingegneria dell'Informazione, University of Parma, Italy

XQLYHUVLW\#RI#SDUPD

DEAS

Devices, Electronic Applications and Sensors Group

Purpose of the work

Thermal design of complex power modules using 3D Finite Element Analysis

Problem of solving equation systems with a very high DOF because of limited computational capabilities of standard PCs

Develop models of electronic components with simplified geometry to use in complex modules

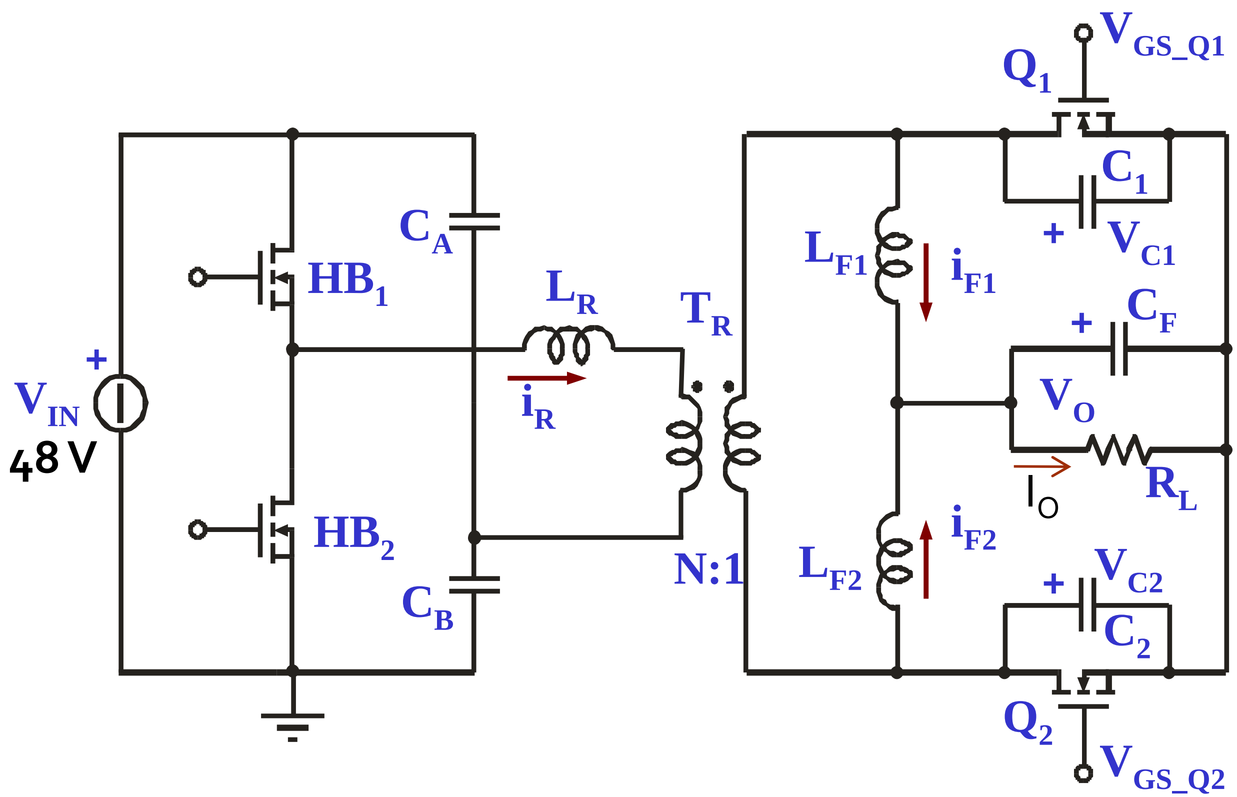
Assembly the simplified models in the power module to verify if they can be used with accuracy in thermal design

Summary

- A DC/DC power converter at high switching frequency
- Thermal design of a power module using FEA
- Modeling workflow
- An example of component study: toroidal inductor operating @ f = 1.8 MHz
- Thermal simplified model of the converter
- Simulations and measurements results

Thermal design of a power module using Finite Element Analysis

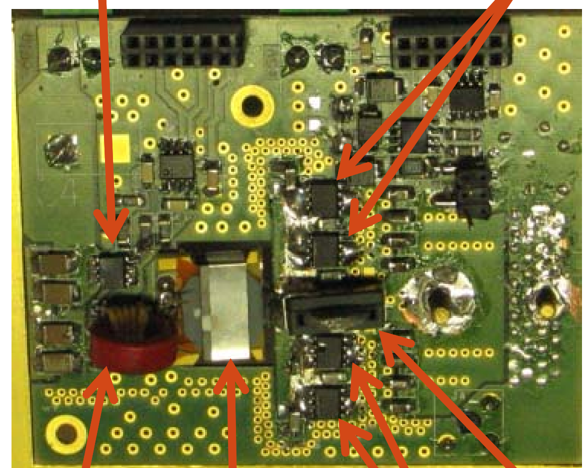
L. Huber, K. Hsu, M. Jovanovic, "**1.8 MHz, 48 V resonant VRM**", IEEE Trans. on Power Electronics, vol. 1, n. 1, 2006



Nominal conditions
V_O = 1.3 V, I_O = 50 A
P_{out} = 65 W

Thermal design of a power module using Finite Element Analysis

Design and prototypes fabrication in cooperation with Department of Information Engineering of University of Padova (Italy)



Nominal conditions

$V_O = 1.3$ V, $I_O = 50$ A

$P_{out} = 65$ W

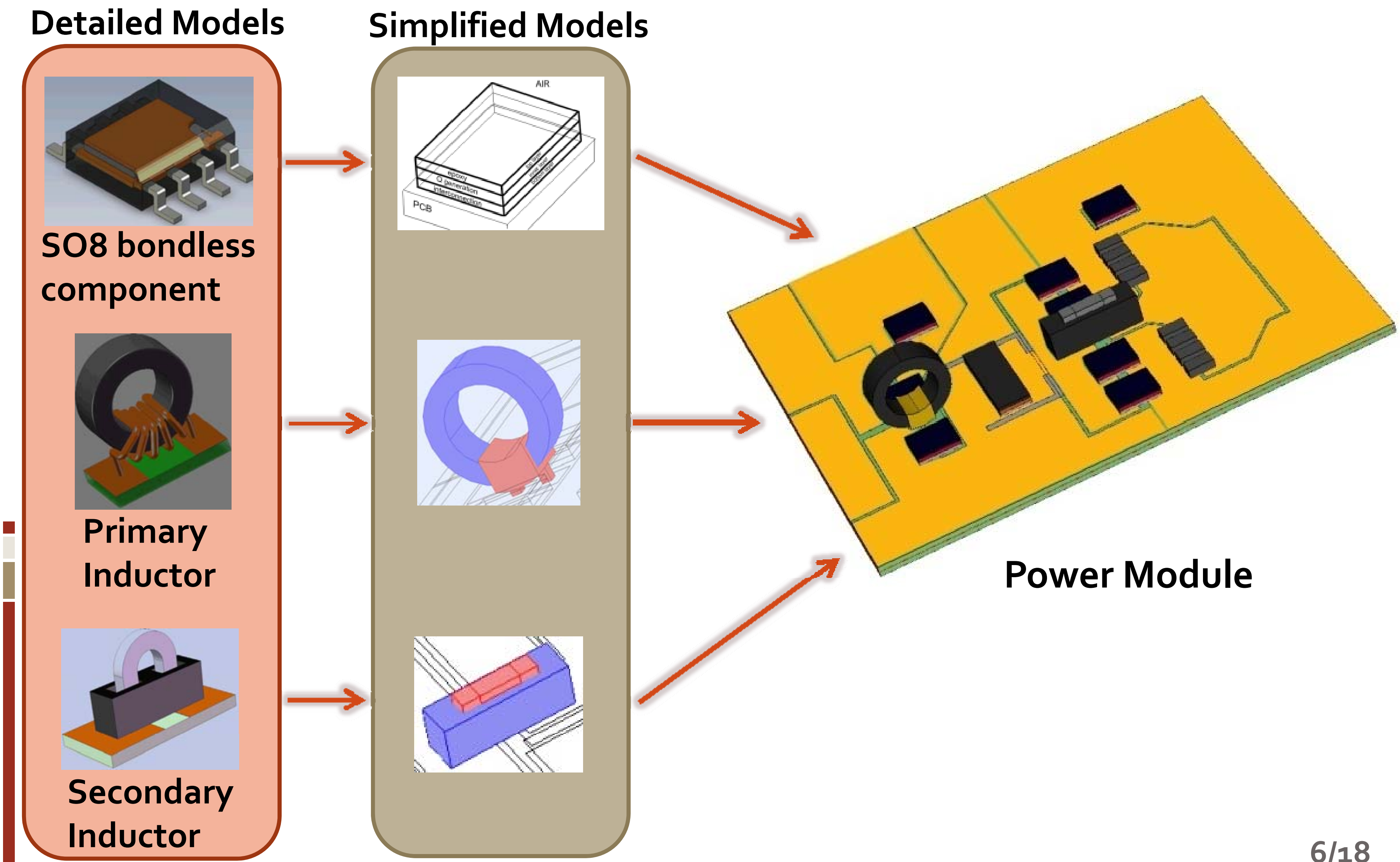
Measured total power losses ~ 12 W

High power density ⇒ FEA useful to design the converter layout

High switching frequency ⇒ significant copper and core losses in magnetic components

Complex geometry ⇒ Large DOF

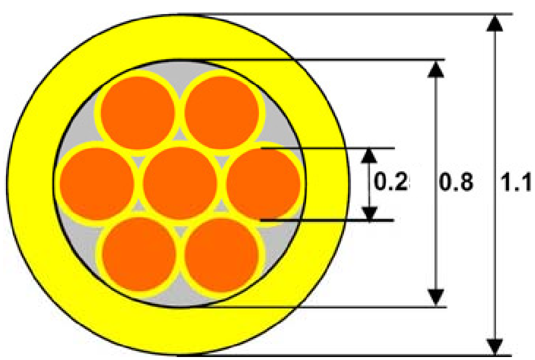
Module modeling workflow



Primary inductor

High f ⇒ Litz wire to avoid cross section area reduction due to skin effect

Schematic section of the Litz wire used

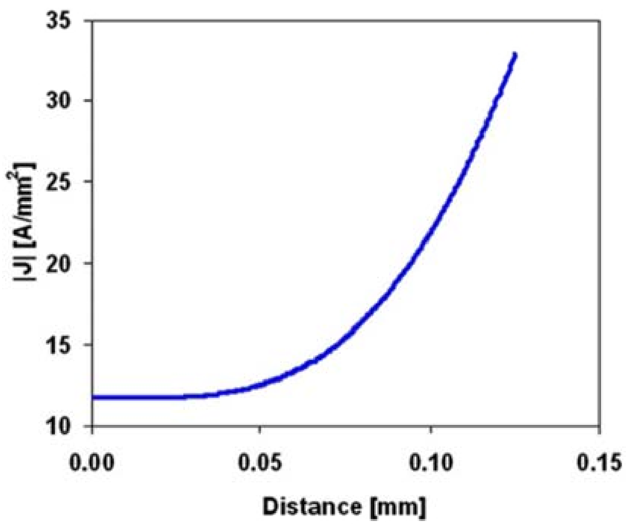


dimensions in mm

High DOF if modeled in 3D

2D simulation of single internal wire of the Litz structure with COMSOL AC Power Electromagnetics Mode

Simulated current density distribution
f = 1.8 MHz, I_{rms} = 4 A (I_{out} = 50 A), T = 20°C



- The distance on the x-axis is measured from the center along a radius.
- The skin depth is about 50 μm.
- The dissipated power density per unit length is 0.17 W/m.

Primary inductor

We designed a 3D wound with a single wire equivalent to the Litz one. We took it into account:

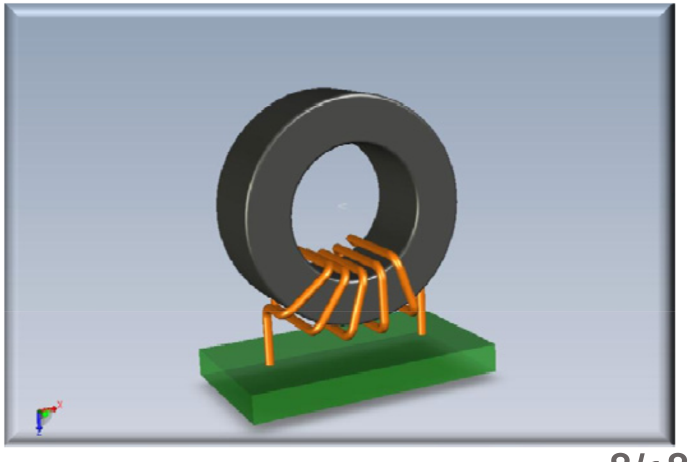
1) The different length of the modeled wire (85 mm instead of 100 mm);
2) The diameter of the designed wire in the model (chosen = 0.9 mm to keep low enough the DOF);
3) The skin effect.

Dissipated Joule Power = *Power Density* x *Length* x n_{wires} = 0.17 × 0.100 × 7 = 0.12 W

assuming this power to be uniformly dissipated in the single wire:

equivalent electrical resistivity = $5.6 \cdot 10^{-8}$ Ωm @ 20°C

(ρ_{Cu} = $1.7 \cdot 10^{-8}$ Ωm @ 20°C)



Primary inductor

Three-step simulation method:

1. Joule effect in the winding using the **Conductive Media DC** application mode;
2. distribution of magnetic flux, eddy currents and the dissipated power density distribution in the core using the **Quasi Static Electromagnetic** application mode;
3. input the dissipated power distributions calculated in the first two steps into the **General Heat Transfer** application mode to evaluate the temperature distribution.

$$\sigma_{Cu\_modified}(T) = \frac{1}{5.6 \cdot 10^{-8}} \cdot \left[1 - 3.9 \cdot 10^{-3}(T - 298)\right]$$

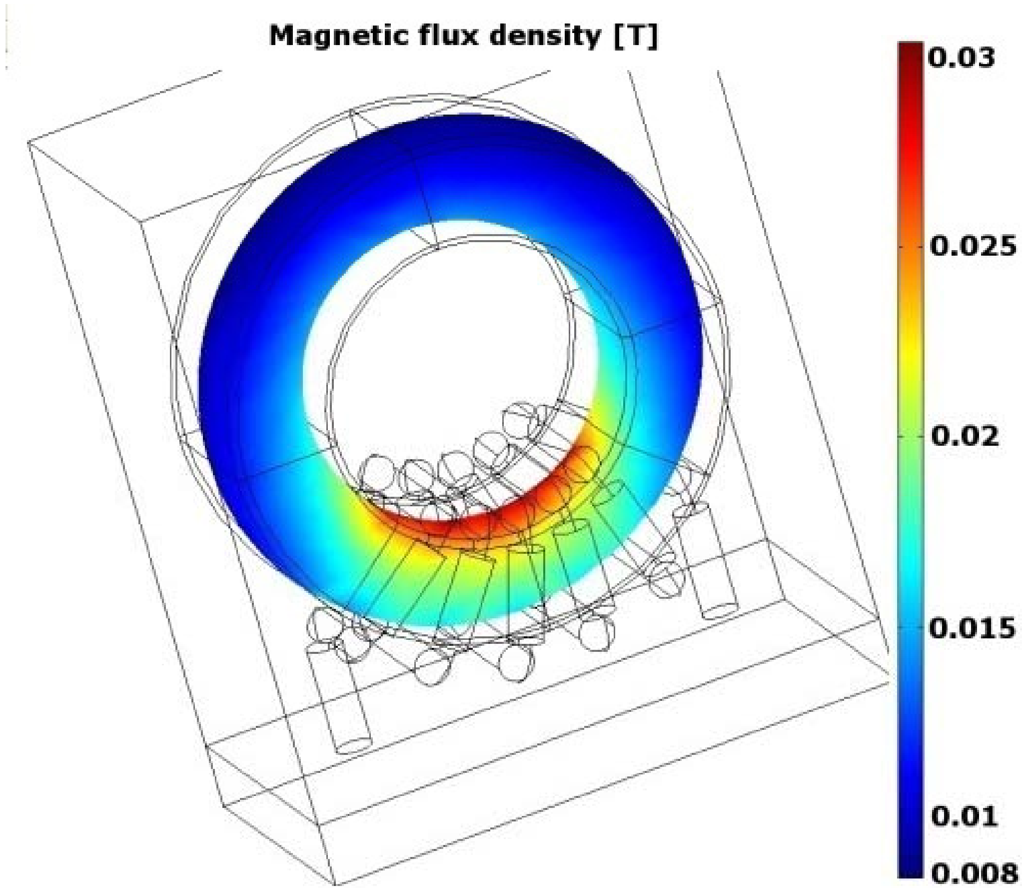
From Micrometals datasheet

$$\mu_{core}(T) = 14 \cdot [1 - 1.5 \cdot 10^{-4}(T - 298)]$$

$$p_s = \frac{f}{\frac{4 \cdot 10^9}{B^3} + \frac{3 \cdot 10^8}{B^{2.3}} + \frac{2.7 \cdot 10^6}{B^{1.65}}} + 1.6 \cdot 10^{-14} f^2 B^2$$

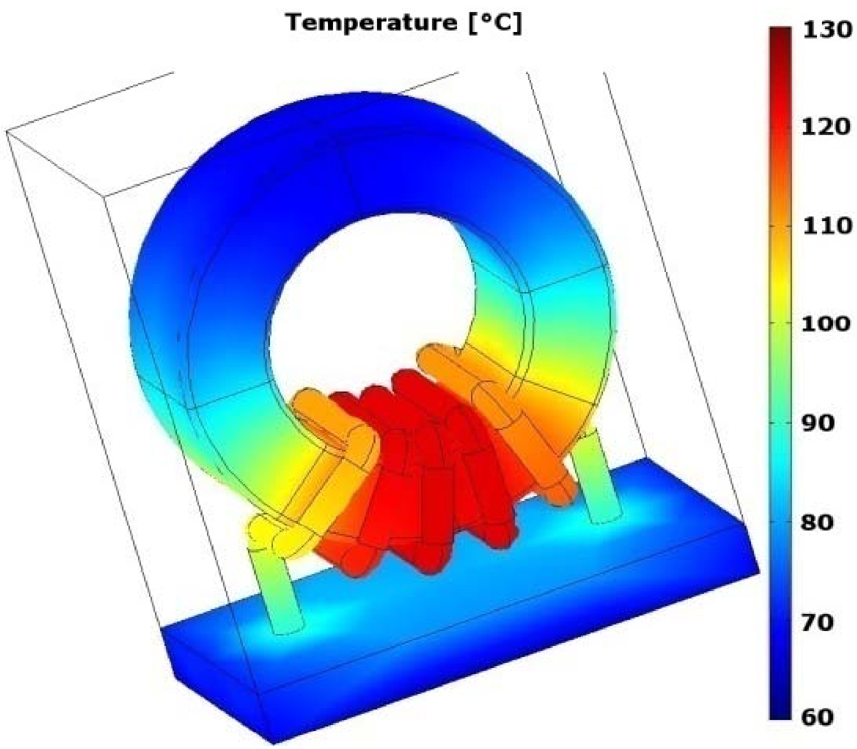
Starting from the three-step simulation we run a self-consistent simulation

Primary inductor



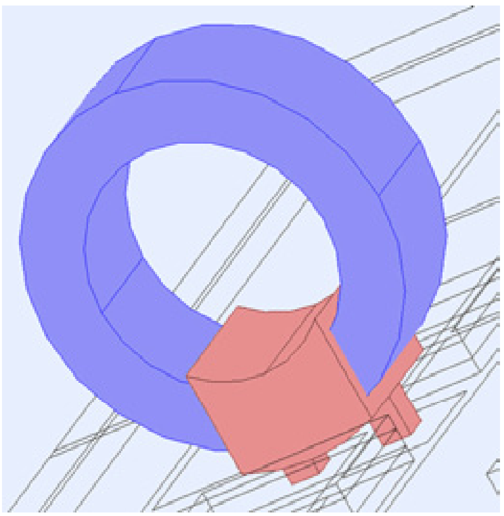
Simulated magnetic flux density in the central section of the resonant inductor core. I_{rms} = 4 A.

Primary inductor



Simulated boundaries temperature distribution of the resonant inductor. I_{rms} = 4 A.

Simplified model

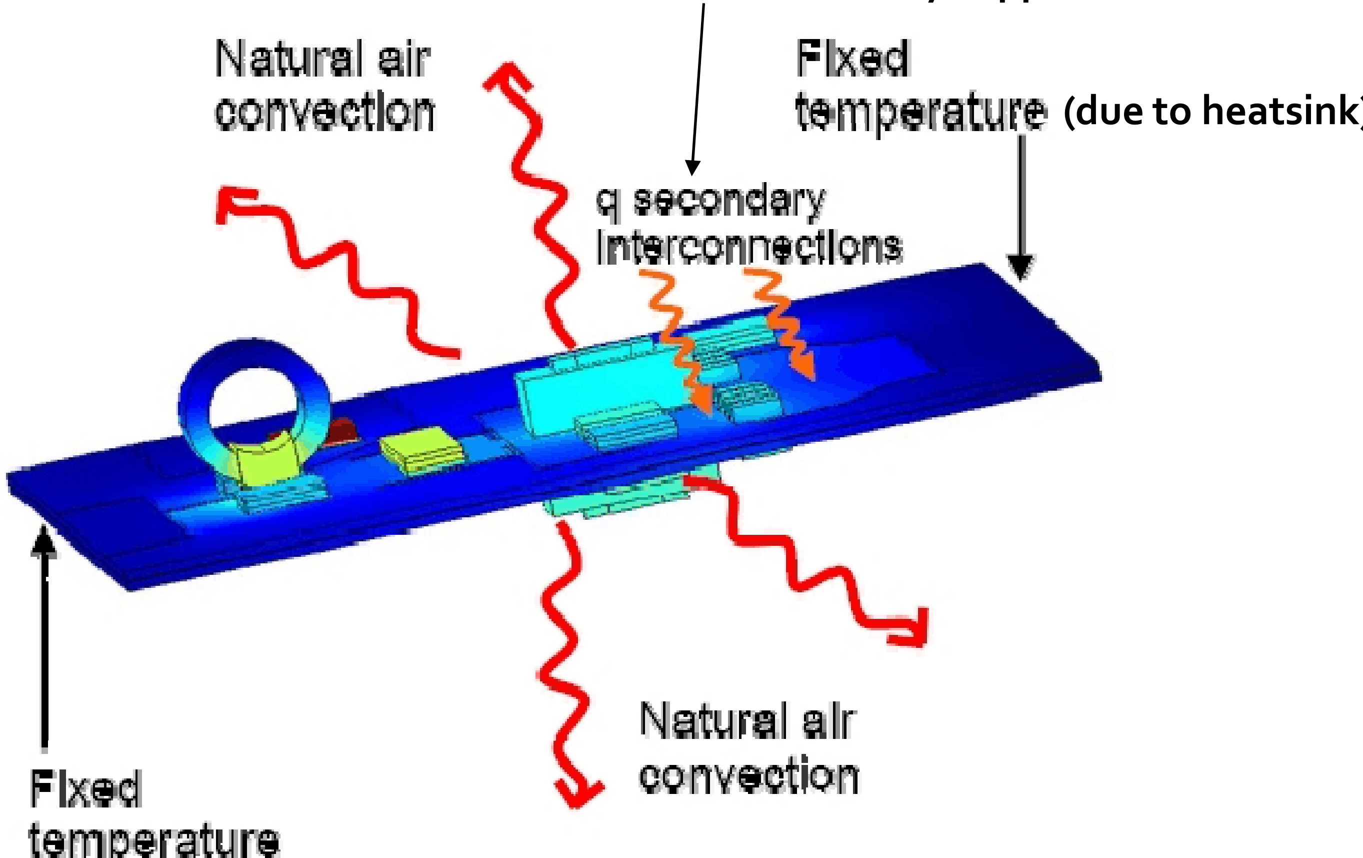


Heat generation only in the sub-domain representing the wire

Module 3D FEM analysis

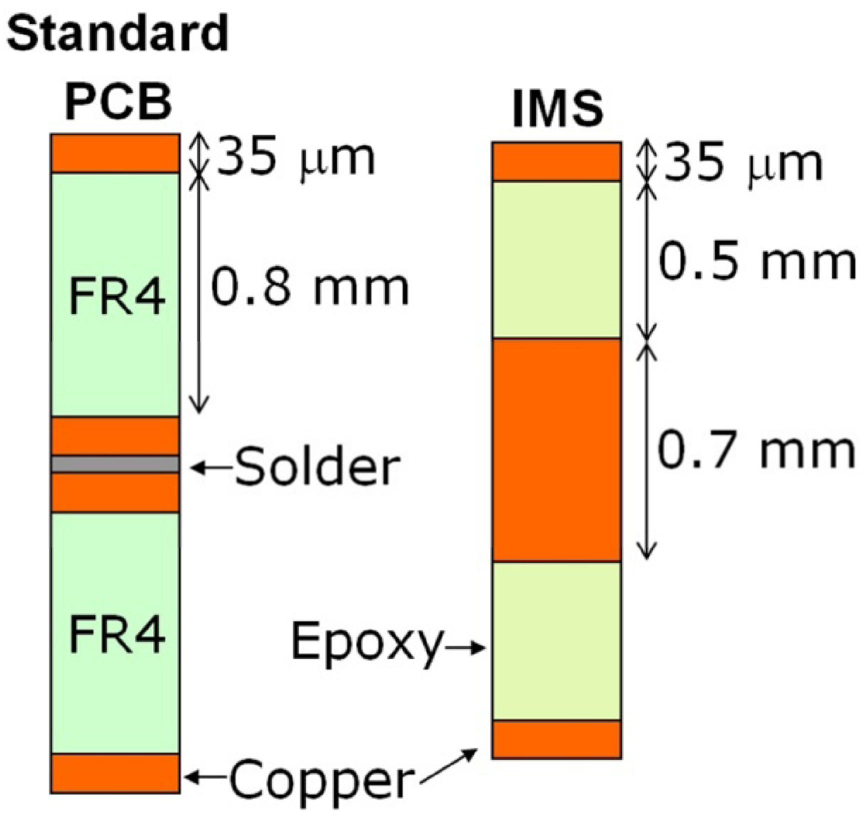
Boundary conditions

Inward heat flux to take into account of the Joule effect in the secondary copper traces



Module 3D FEM analysis

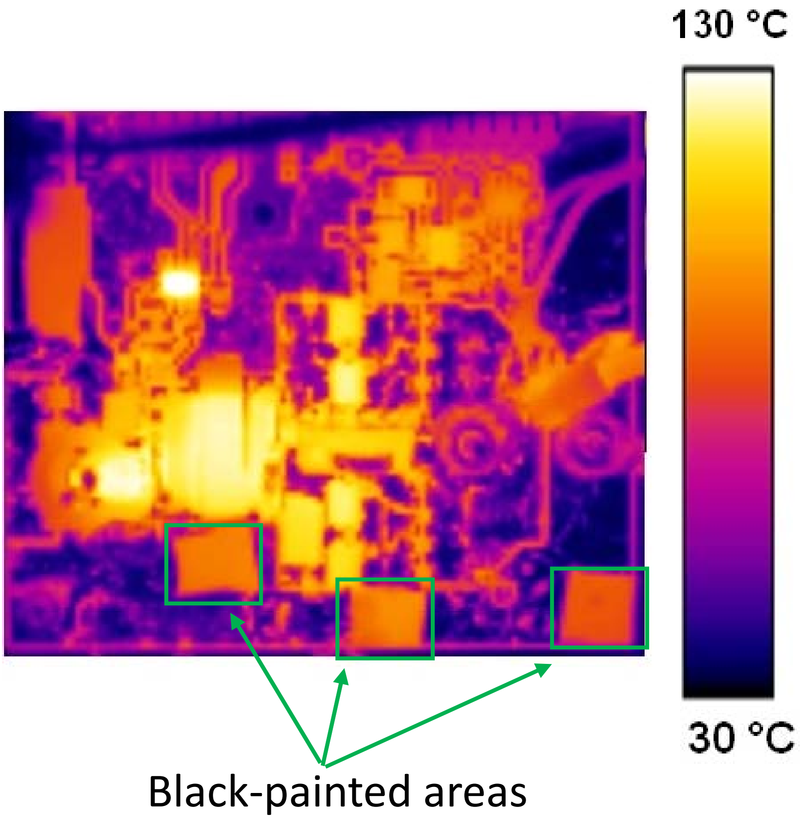
Steady state analysis with Comsol Multiphysics
of two solutions with the same components layout but different substrate:



Standard PCB solution results

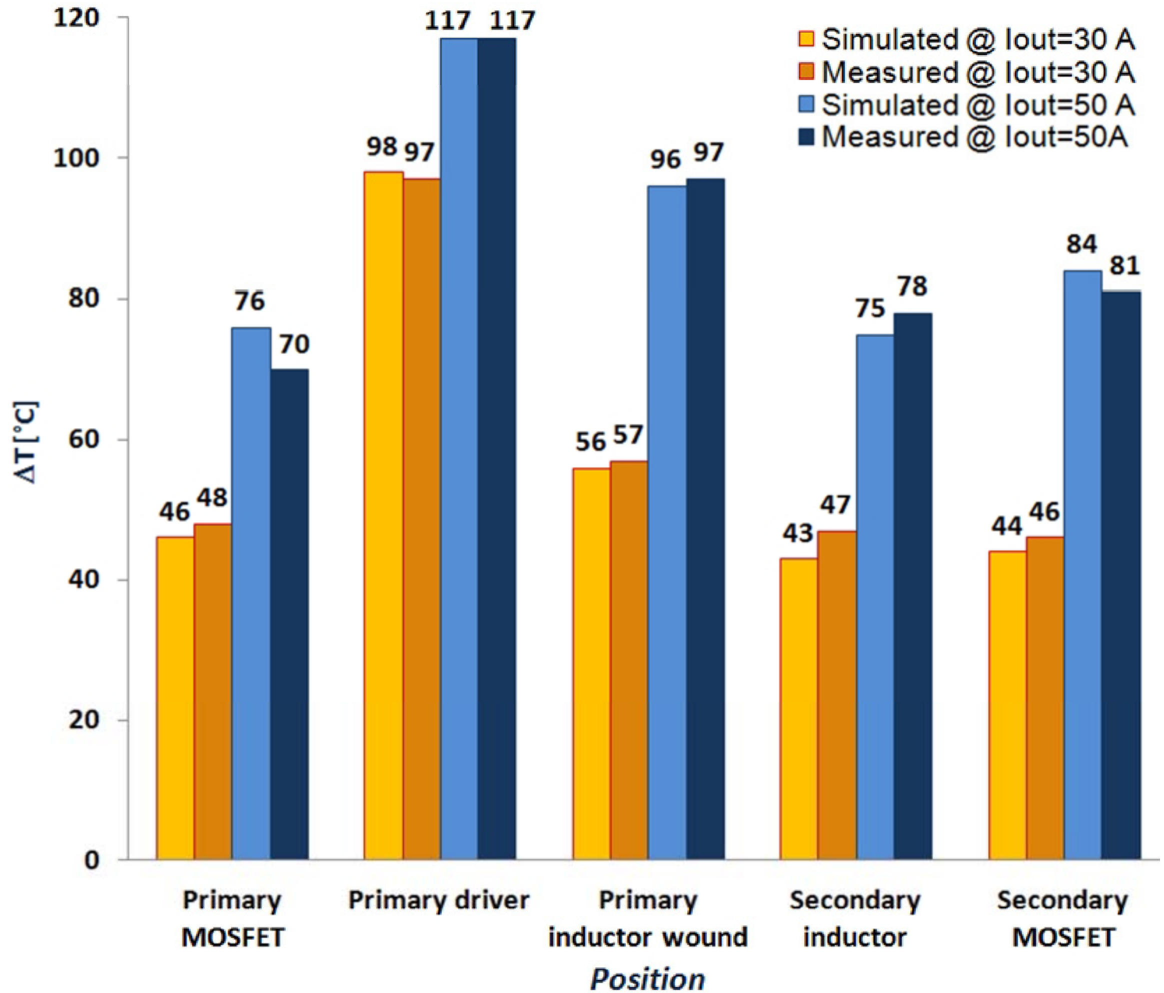
@ I_{out} = 50 A

COMSOL 3D thermal simulation



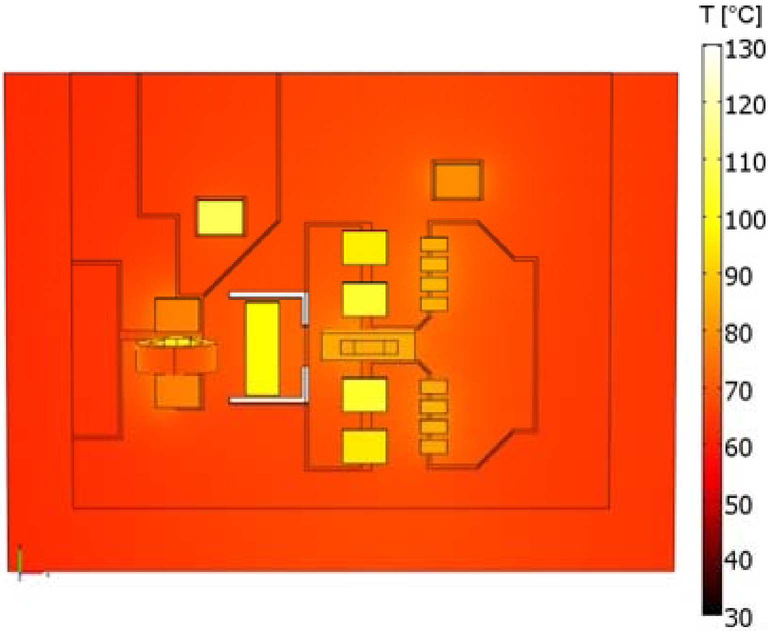
IR camera image

Standard PCB solution results

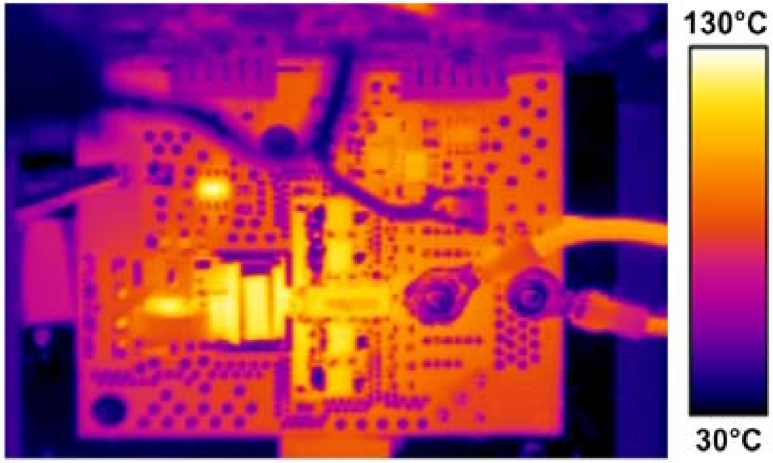


IMS solution results

@ I_{out} = 50 A



COMSOL 3D thermal simulation



IR camera image

Results comparison

Component (test point)	ΔT [°C] (FR4)		ΔT [°C] (IMS)	
	Measured	Simulated	Measured	Simulated
Primary inductor wound	97	96 (-1%)	80	71 (-11%)
Primary inductor core	42	44 (+5%)	42	42 (<1%)
Transformer	95	99 (+4%)	82	73 (-11%)
Secondary inductor	78	75 (-4%)	70	61 (-13%)

Used for parameter fitting

Conclusions

- This work focused on the thermal modeling of power converter boards.
- In order to forecast the critical component temperature a FE model of the whole converter has been developed.
- The accurate FE model of some components is used as reference to build simplified models that can be used in the FE thermal simulation of the whole converter board.
- The results obtained are encouraging, and indicate that this approach can be a valuable tool providing designers with a way to evaluate the interaction between electrical, magnetic and thermal effects while choosing the proper components, materials and layout for the converter.