Design of Mechatronic Devices by Electro-Thermal FE Analysis Coupled to CFD Analysis

Jacques Marchesini, Reinhard Helfrich, Manuel Henner

INTES France ; INTES GmbH, Germany ; VALEO, France

1 Introduction

In the engine cooling module, the thermal design of the electronic power module which controls the fan rotation speed by a prescribed voltage needs the induced heat sources from electrical current and also the dissipation effects from heat convection to surrounding fluid flow. So, to achieve an accurate thermal analysis, a multi-physics simulation is required with the consideration of the electrical current distribution by a Finite Element (FE) solution and the surrounding fluid behaviour by a Computational Fluid Dynamics (CFD) analysis.

2 Mechatronic device

2.1 Engine cooling module

The engine cooling module [Fig. 1] allows the evacuation of the heat generated by the car engine. This function needs the rotation of a fan to refresh the radiator. The control of the fan rotational speed is managed by the electronic power module which transfers up to 600 Watt of power toward the electric fan motor. A part of this power is dissipated by Joule effect in the electronic power module.

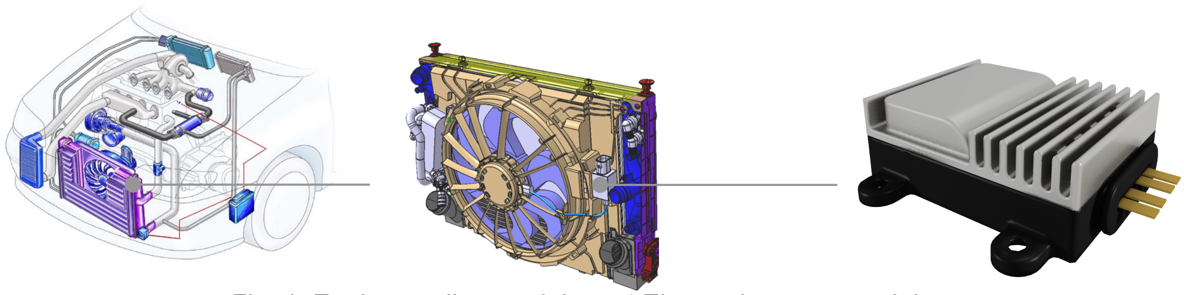

Fig. 1: Engine cooling module and *Electronic power module*

2.2 Electronic power module

The electronic power module (PWM) [Fig. 1] is mainly composed by a housing and the electronic card. The housing is a plastic box closed by an aluminium dissipater which surrounds the electronic card. The electronic card contains the electrical power circuit (thyristors, diodes…) which transfers a current up to 50A, and the electronic command circuit which controls the commutation of thyristors in order to prescribe the average voltage to be delivered to the fan motor.
The design of the PWM needs the accurate knowledge of the temperature distribution in order to optimize the shape of the aluminium dissipater as well as the choice of the quality class of electric components, which have a direct impact to the cost price.

3 Multiphysics simulation

The simulation uses a detailed 3D finite-element model of the PWM and starts with a steady electric current analysis, in order to generate the volumetric heat produced by Joule effect from the current density distribution. In the same run, this heat quantity is used directly as distributed thermal load for the subsequent temperature analysis. During this thermal analysis, the FE model is coupled with a finite volume model which represents the surrounding fluid and provides the thermal boundary conditions to the solid domain. The figure 2 shows the multiphysics simulation scheme:

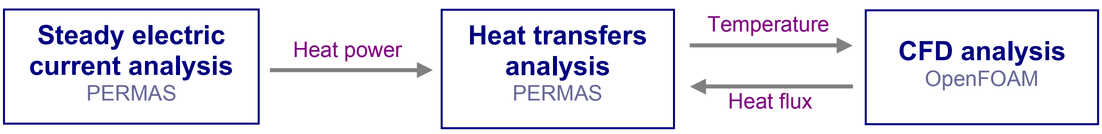

Fig. 2: Multiphysics simulation scheme

3.1.1 The solid model

Only the electric power circuit is considered for the steady electric current analysis [Fig. 3]. This circuit part is mainly modelled by solid elements. Surface elements and line elements are used also for the representation of very thin copper structures or P-N junctions in easy way. Incompatible meshes allow to assembly all parts in fast way. The load is a prescribed input voltage. This analysis is done with the F.E. solver PERMAS [1] and generates the volumetric heat produced by Joule effect from the current density distribution.
The mesh used for the electric analysis is also used for the heat transfers analysis which simplifies the modelling process: F.E. elements have a potential scalar dof and a temperature dof at nodes. The solid model has a thermal conduction behaviour and the boundary conditions are the prescribed heat fluxes which are given by the surrounding fluid model. Thermal loads are given by the previous steady electric current analysis.

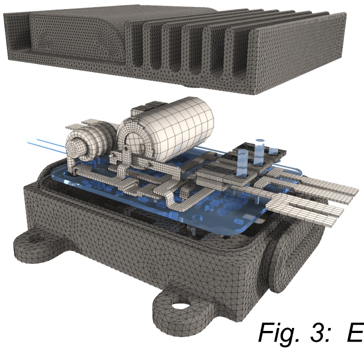
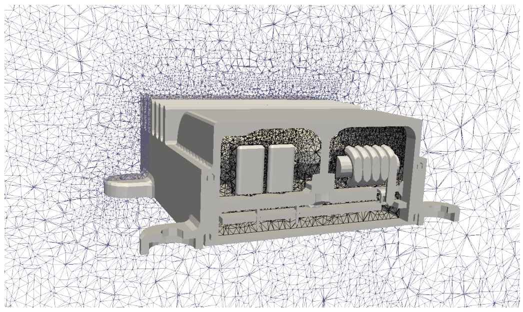

Fig. 3: Electro-thermal solid mesh and Fluid domains

3.1.2 The fluid model

The fluid domain [fig. 3] is divided in two regions: the internal cavity which is considered to be in natural convection and the outside fluid which is considered to be in forced convection (from the fan rotation). This model is managed by the solver *'buoyantSimpleFoam'* from the C.F.D. software OpenFOAM [2]. A coupled process in bidirectional way is established with the F.E. solver PERMAS in order to exchange the wall temperatures and the heat fluxes.

3.1.3 The simulation results

The steady electric current analysis shows especially the current density [Fig. 4] which indicates the possible hot-points. The subsequent coupled thermal analysis gives the temperature distribution [Fig. 4] and paths of heat transfers inside the internal fluid cavity. This indicates also the efficiency of the dissipater and the effect of fluid flow direction.

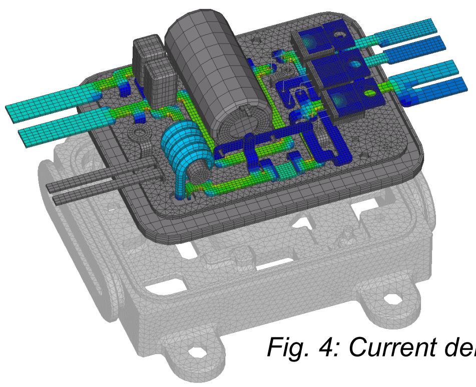
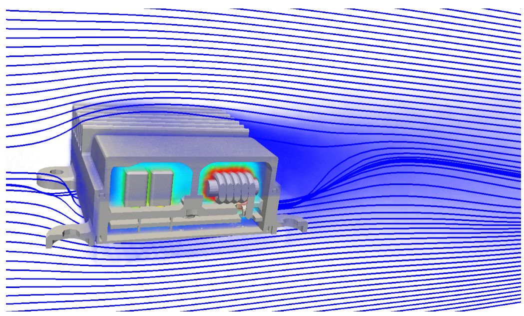

Fig. 4: Current density and Fluid temperature distribution

4 Conclusion

An efficient coupled process between an electric-thermal solid analysis and a fluid flow analysis, combined with advanced modelling features like incompatible meshes and unified models allows the simulation of complex models like the electric power module in affordable time.
Such multiphysics simulations give accurate information in the daily work of designers and become closer to their needs. Several variants (like the input voltage, the dissipater shape, the fluid flow direction…) could be easily verified.

5 References

[1] PERMAS User's Reference Manual, INTES Publication UM 450, Stuttgart 2012, www.intes.de.
[2] OpenFOAM V2.1.0 User Guide Documentation, OpenCFD Ltd., www.openfoam.org.