Available online at www.sciencedirect.com
ScienceDirect
Comput. Methods Appl. Mech. Engrg. 282 (2014) 239–268
Computer methods in applied mechanics and engineering
www.elsevier.com/locate/cma

A multiscale thermo-fluid computational model for a two-phase cooling system

Riccardo Sacco[a,*], Lucia Carichino[b], Carlo de Falco[d,e], Maurizio Verri[a], Francesco Agostini[c], Thomas Gradinger[c]

[a] *Dipartimento di Matematica, Politecnico di Milano, Piazza L. da Vinci 32, 20133 Milano, Italy*
[b] *IUPUI Department of Mathematical Sciences, 402 N. Blackford St., LD 270 E Indianapolis, IN 46202-3267, United States*
[c] *ABB Switzerland Ltd., Corporate Research, Segelhofstrasse 1K 5405, Baden-Dättwil, Switzerland*
[d] *MOX Modeling and Scientific Computing, Dipartimento di Matematica, Politecnico di Milano, Piazza L. da Vinci 32, 20133 Milano, Italy*
[e] *CEN Centro Europeo di Nanomedicina, Piazza L. da Vinci 32, 20133 Milano, Italy*

Received 4 April 2014; received in revised form 10 July 2014; accepted 10 August 2014
Available online 22 August 2014

Highlights

- We model a novel two-phase cooling system for power electronics.
- The two-phase coolant flows in a network of pipes.
- Dissipated heat is removed through air convection.
- Multiscale techniques are used for model complexity reduction.
- Approximation with monotone and flux-conservative finite elements.

Abstract

In this paper, we describe a mathematical model and a numerical simulation method for the condenser component of a novel two-phase thermosiphon cooling system for power electronics applications. The condenser consists of a set of roll-bonded vertically mounted fins among which air flows by either natural or forced convection. In order to deepen the understanding of the mechanisms that determine the performance of the condenser and to facilitate the further optimization of its industrial design, a multiscale approach is developed to reduce as much as possible the complexity of the simulation code while maintaining reasonable predictive accuracy. To this end, heat diffusion in the fins and its convective transport in air are modeled as 2D processes while the flow of the two-phase coolant within the fins is modeled as a 1D network of pipes. For the numerical solution of the resulting equations, a Dual Mixed-Finite Volume scheme with Exponential Fitting stabilization is used for 2D heat diffusion and convection while a Primal Mixed Finite Element discretization method with upwind stabilization is used for the 1D coolant flow. The mathematical model and the numerical method are validated through extensive simulations of realistic device structures which prove to be in excellent agreement with available experimental data.

Keywords: Cooling systems; Fluid-dynamics; Two-phase flow; Homogeneous flow; Multiscale modeling; Numerical simulation

* Corresponding author. Tel.: +39 02 2399 4540; fax: +39 02 2399 4513.
E-mail address: riccardo.sacco@polimi.it (R. Sacco).
URL: http://www1.mate.polimi.it/~ricsac/ (R. Sacco).

http://dx.doi.org/10.1016/j.cma.2014.08.003
0045-7825/

1. Introduction and motivation

Ever since the early 1980s the increasing growth of new technologies and applications has been shifting scientific interest on power electronics. In such wide-range industrial context, the necessity to develop devices with a high power dissipation per unit volume has justified the need of advanced cooling systems capable to prevent excessive temperature increase and consequent device failure. Conventional cooling procedures exploit convection heat transfer between a fluid in motion and a bounding surface at different temperatures. Typical examples are water-cooled and air-cooled systems, widely used in power electronics applications. A different approach to cooling is represented by the two-phase thermosiphon device whose functioning principle is schematically illustrated in Fig. 1 and whose structure is shown in Fig. 2(a).

This kind of device consists of an evaporator, attached to the device requiring cooling, and a condenser made up of a stack of fins among which air is allowed to flow. The evaporator and the condenser are connected by a pipe in which a mixture of liquid and vapor phases is flowing. The heat generated by an electronic device in contact with the evaporator is collected by means of an evaporating fluid. The vapor phase fluid, rising in the pipe, passes through the condenser where it returns to the liquid phase. As no pumps are needed to move the refrigerant fluid from the evaporator to the condenser, the resulting thermodynamical efficiency of two-phase cooling systems is remarkably superior to that of water-cooled or air-cooled systems (see [3]). In order to deepen our understanding of the mechanisms that determine the performance of a two-phase thermosiphon cooler device and to facilitate the further optimization of its design, in the present research we focus on the study of the condenser subsystem (see Fig. 2(b)), for which we develop a multiscale mathematical model that is implemented in a numerical simulation code. As computational efficiency is a stringent requirement in industrial design and optimization procedures, model complexity is suitably reduced through the adoption of physically sound consistent assumptions that allow us to end up with a system of nonlinearly coupled 2D PDEs for the air and panel temperatures, and 1D equations within the network of pipes distributed in each fin for the refrigerant fluid flow.

Another important constraint is represented by the ability of the computational method to reproduce on the discrete level important physical features characterizing the problem at hand, such as mass and flux conservation, and its robustness in the presence of dominating convective flow regimes. These requirements are here satisfied by the introduction of a stabilized mixed finite element scheme on quadrilateral grids that automatically provides the desired inter-element flux conservation and upwinding through the use of suitable quadrature rules for the mass flux matrix and convection term. The resulting discrete method has also an immediate interpretation in terms of finite volume formulation which allows a compact implementation of the scheme that highly improves the overall efficiency of the computer-aided design procedure.

A final issue of critical importance in the development of a reliable computational tool for use in industrial design is model calibration and validation. Model calibration is properly addressed by supplying the parameter setting in the equation system with suitable *empirical correlations*, that are functional relations between two or more physical variables, usually obtained by means of a series of experimental tests. In common engineering practice, correlations are widely used because they allow to account for complex physical phenomena in a simple and synthetic manner, albeit their applicability is clearly restricted to a specific admissible range of parameter values. Model validation is carried out through extensive numerical simulations of the two-phase condenser under realistic working conditions.

An outline of the article is as follows. Section 2.1 describes the two-dimensional model for heat convection in air and heat diffusion in the panel whose derivation from the corresponding full 3D model is outlined in the Appendix. The simplified geometrical representation of the coolant-filled channel and the system of 1D equations describing the flow within it are dealt with in Section 2.2. Section 3 discusses the decoupled iterative algorithm used to solve the complete model while Sections 4 and 5 are devoted to the discussion of the discretization techniques adopted to treat each differential subsystems arising from system linearization. Finally, in Section 6 simulation results are presented and discussed and in Section 7 conclusions are drawn and possible future research directions are addressed.

2. Mathematical models

In this section we describe the mathematical model on which our numerical simulation tool for the condenser is based. The equations for heat convection in air and heat diffusion in the panel wall are presented in Section 2.1, while the model for the two-phase flow in the channel is in Section 2.2.