International Journal of Heat and Mass Transfer 64 (2013) 759–772

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Development of heat sink device by using topology optimization

Adriano A. Koga [a], Edson Comini C. Lopes [b], Helcio F. Villa Nova [c], Cícero R. de Lima [b,*], Emílio Carlos Nelli Silva [a]

[a] Department of Mechatronics and Mechanical Systems Engineering, Escola Politécnica da Universidade de São Paulo - Avenida Professor Mello Moraes, 2231, CEP 05508-900, São Paulo, SP, Brazil

[b] School of Engineering, Modeling and Applied Sciences, UFABC, Av. dos Estados, 5001, CEP 09210-580, Santo André, SP, Brazil

[c] Institute of Mechanical Engineering, Federal University of Itajubá, CEP: 37500 903, Itajubá, MG, Brazil

ARTICLE INFO

Article history:
Received 3 October 2012
Received in revised form 18 April 2013
Accepted 3 May 2013
Available online 4 June 2013

Keywords:
Heat sink
Stokes flow
Heat transfer maximization
Pressure drop minimization
Topology optimization
CFD

ABSTRACT

Small scale fluid flow systems have been studied for various applications, such as chemical reagent dosages and cooling devices of compact electronic components. This work proposes to present the complete cycle development of an optimized heat sink designed by using Topology Optimization Method (TOM) for best performance, including minimization of pressure drop in fluid flow and maximization of heat dissipation effects, aiming small scale applications. The TOM is applied to a domain, to obtain an optimized channel topology, according to a given multi-objective function that combines pressure drop minimization and heat transfer maximization. Stokes flow hypothesis is adopted. Moreover, both conduction and forced convection effects are included in the steady-state heat transfer model. The topology optimization procedure combines the Finite Element Method (to carry out the physical analysis) with Sequential Linear Programming (as the optimization algorithm). Two-dimensional topology optimization results of channel layouts obtained for a heat sink design are presented as example to illustrate the design methodology. 3D computational simulations and prototype manufacturing have been carried out to validate the proposed design methodology.

1. Introduction

Channel flow systems have been largely used, from nano to large scale, in various applications. Small scale fluid flow systems found in nano/microchannel devices have been studied over the last years as potential devices to be applied for chemical reagent dosages and for cooling compact electronic components, for example. Focusing on these small scale devices, we can highlight applications such as fluid mixers [1], pharmaceutic industry analysis [2], sensors and biological applications [3], and lab-on-a-chip systems [4,5]. Weibel et al. [6] have applied microchannel and valves for bio-chemical analysis, while Beebe et al. [7] have studied an application of a microchannel network to biological studies and analyses. Psaltis et al. [8] have applied microchannel devices to control fluid chemical composition of a given fluid, which affects its optical properties (polarization, refraction, etc.) and therefore, allowing the control of an optical based device.

Other notorious application of the small scale fluid flow systems are microchannel heat sinks, which make efficient the cooling of compact and powerful electronic devices. Microchannel heat sink design represents an important field of study, and it has to evolve to pair up with the crescent microelectronics progresses. As first demonstrated by Tuckerman and Pease [9], reducing of hydraulic diameter of the channel can provide large convective heat transfer coefficient, and small mass and volume for a heat sink. Since then, microchannel technology has been studied over the last three decades as solution to increase cooling efficiency of microelectronic devices, which produces large amount of heat in a small area. Nevertheless, in order to achieve better heat sink designs, it is crucial that these cooling systems have a very efficient fluid flow channel, which can be obtained by minimizing pressure drops along its extension [10–12]. Thus, given the great potential applications inherent to microchannel devices, it is natural to aim for an optimization study in this field.

Design optimization studies have been investigated by many researchers to determine the geometric dimensions of microchannels that give optimum performance for a heat sink device [13–17]. In general, these studies have been carried out by using numerical analyses of the fluid flow and heat transfer in microchannels together with a parametric optimization scheme to find the optimal cross-section parameters of the microchannel heat sink.

However, the basics on the application of optimization methods in fluid flow problems was given by Pironneau [18], who considers shape optimization of minimum drag profiles under a Stokes flow.

* Corresponding author. Tel.: +55 11 4996 0107; fax: +55 11 4996 0089.
E-mail addresses: helcio.villanova@gmail.com (H.F. Villa Nova), cicero.lima@ufabc.edu.br (C.R.d. Lima), ecnsilva@usp.br (E.C.N. Silva).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.05.007

Later, Borrvall and Petersson [19] gave the first steps on the application of the Topology Optimization Method (TOM) for minimum power dissipation in fluid flow channels, comparing its results with the previous Pironneau's results. The main advantage of the TOM is the capacity of distributing material freely, inside a design domain, and systematically searching for the best-performance topology [20]. Thus, by applying this method, a "not-so-intuitive" optimal solution can be achieved with no need of proposing a "pre-structured" initial guess, as needed in parametric and shape optimization. Gersborg-Hansen et al. [21], and Guest and Prévost [22] have extended the TOM for other applications, such as non-linear effects and a broad Reynolds response, considering the full Navier–Stokes equations. Evgrafov [23] has studied the effects of compressibility over the TOM solutions, and Pingen and Maute [24] have studied the TOM application in non-Newtonian fluids. Aage et al. [25] have extended the application of the TOM to large scale Stokes flow (3D) problems, to obtain finer geometry details of the topology optimization result.

Another field of interest is related to the TOM application in heat transfer problems, such as heat conduction problems [26], conduction–convection problems [27], and nano-scale heat transfer problems [28]. Donoso and Sigmund [29] have applied TOM to analyze multiple physics design problems that can be described by Poisson's equation, giving new physical insight in the various optimization problems, such as heat conduction and ground-water flow. Additionally, Takezawa and Kitamura [30] have carried out a topology optimization methodology for the geometrical design of thermoelectric generators, by introducing an analytical model that mimics the closed circuit for the thermoelectric problem.

However, few works have explored TOM applied to fluid flow and heat transfer coupled problems. We can cite the work of Dede [31], who has discussed the use of a commercial finite element software (COMSOL) and MATLAB script for topology optimization of heat transfer and fluid flow multiphysics problems. However, more details could be given about the physical input values (velocity, temperature, etc.) considered in his results. Okkels et al. [32] have also added the temperature field in the topology optimization Navier–Stokes flow problem, for a single microfluidic system in which an inner square is cooled by the flow of a cooling liquid. However, more details could be given about the defined objective function formulation when the temperature field has been included in their topology optimization problem, and how the material model relates the design variable with the physical properties of the material (thermal conductivity, specific heat, and material density) in the heat dissipation problem. Finally, the work developed by Yoon [33] also studies the heat dissipation effects over the structure design, by using TOM. In this case, a thermal compliance minimization [20] is considered as cost function.

However, none of these works has considered to study the complete cycle of development (Fig. 1) of heat sink devices designed by using topology optimization. Thus, the objective of this work is to propose a complete cycle of development of a heat sink device with higher thermal dissipation and lower pressure drop designed by using TOM. The implemented design methodology based on TOM consists of obtaining an optimized solution for channel layouts to attend a multi-objective optimization problem which involves minimization of the pressure drop of fluid flow, and maximization of the heat dissipation along the channel. 3D computational simulations and experimental characterization of a manufactured prototype are carried out to validate the proposed design methodology.

Additionally, this work focuses in small scale applications, by aiming water based devices, at low velocities and low Reynolds (laminar flow), where the inertial forces are negligible, under steady-state, resulting in a Stokes flow model. Thus, the model utilized in this work considers incompressible flow of viscous Newtonian fluids (Stokes flow). The heat transfer model considers both conduction and forced convection models, in steady-state and it considers that the physical properties are constant with temperature.

This paper is organized as follows. In Section 2, the mathematical formulation of the fluid flow and heat transfer models are introduced. In Section 3, it is shown the basic topology optimization concepts and the main characteristics relevant to this application, including the topology optimization problem and its numerical implementation. In Section 4, the computational modeling and the experimental apparatus applied for prototype characterization are described. In Section 5, it is presented two-dimensional topology optimization results, as well as the 3D computational simulation and experimental characterization results of a manufactured heat sink prototype. Finally, in Section 6, the results are discussed and some concluding remarks are given.

2. Problem modeling

According to the assumptions made in the first section of this paper, the mathematical formulation of fluid flow and heat transfer models adopted in this work are discussed here.

2.1. Fluid flow

The full Navier–Stokes equations [34], described by the momentum and continuity equations, are presented as follows:

$$\rho_m\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u}\right) = -\nabla p + \mu\nabla^2\mathbf{u} + \mathbf{f} \tag{1}$$

$$\frac{\partial \rho_m}{\partial t} + \nabla\cdot(\rho_m\mathbf{u}) = 0 \tag{2}$$

where, ρ_m is the fluid density, $\mathbf{u}$ and p are the velocity and pressure fields, respectively, μ is the fluid dynamic viscosity and $\mathbf{f}$ the fluid body forces.

Considering a steady-state flow of an incompressible Newtonian fluid at low Reynolds, where inertial effects are negligible, the Stokes equations are retained, given by [19]:

$$\begin{aligned} &-\mu\nabla^2\mathbf{u} + \nabla p = \mathbf{f} \\ &\nabla\cdot\mathbf{u} = 0 \end{aligned} \tag{3}$$

For describing solid regions, a mixed model for fluid flow through a porous medium approach, named Darcy's law [22], has been applied:

$$\begin{aligned} &\alpha\mathbf{u} = (\nabla p - \mathbf{f}) \\ &\nabla\cdot\mathbf{u} = 0 \end{aligned} \tag{4}$$

where α is the inverse permeability of the porous medium.

The main idea behind the application of a porous media model is to control the material permeability in such a way that a solid behavior can be simulated by setting the material permeability equal to zero. Thus, in the limit, the fluid velocity in this region goes to zero.

The Stokes flow and the Darcy's law are combined by using the Brinkman equations [19,22], which allow the use of both models in a single system equation, as follows:

$$\begin{aligned} &\mu\nabla^2\mathbf{u} + \alpha\mathbf{u} = \nabla p - \mathbf{f} \\ &\nabla\cdot\mathbf{u} = 0 \end{aligned} \tag{5}$$

The Brinkman equations may represent both solid and fluid materials, in a specific region of the domain, which might be constantly changed during the optimization process. Mathematically, the α term allows the creation of a zero-permeability region, acting as a velocity absorption term, which sets velocity equal to zero. For