Asociación Argentina de Mecánica Computacional

Mecánica Computacional Vol XXIX, págs. 8307-8317 (artículo completo)
Eduardo Dvorkin, Marcela Goldschmit, Mario Storti (Eds.)
Buenos Aires, Argentina, 15-18 Noviembre 2010

MICROCHANNEL HEAT SINK DESIGN BASED ON TOPOLOGY OPTIMIZATION

Cícero R. de Lima[a], Adriano A. Koga[b], and Emílio C. N. Silva[b]

[a]*UFABC – School of Engineering, Modeling and Applied Sciences, Av. dos Estados, 5001, Santo André, Brazil, cicero.lima@ufabc.edu.br*

[b]*University of São Paulo – Department of Mechatronic Engineering and Mechanical Systems, Av. Prof. Mello Moraes, 2231, São Paulo, Brazil*

Keywords: microchannels, topology optimization, finite element, computational simulation.

Abstract. Microchannel heat sink design consists in an innovative technology which has been studied as alternative to increase cooling efficiency of small electronic devices, such as high-end microprocessors of CPUs. These electronics devices dissipate a large amount of heat, which requires very efficient cooling systems. Microchannels constructed on a conductivity body allow obtaining an efficient heat sink design having better thermal dissipation with small mass and volume, and large convective heat transfer coefficient, and, thus, suitable for cooling compact areas of small electronic devices. Thus, the main objective of this work is the study of a methodology to develop a microchannel heat sink design through the application of the Topology Optimization Method, which allows the distribution of a limited amount of material, inside a given design domain, in order to obtain an optimized system design. This method combines the Finite Element Method (FEM) and Sequential Linear Programming (SLP) to find, systematically, an optimized layout design for microchannels in heat sinks. Essentially, the topology optimization problem applied to channel fluid flow consists of determining which points of a given design domain (small heat sink) should be fluid, and which points should be solid to satisfy a multi-objective function that maximizes the heat dissipation, with minimum pressure drop. In this proposed methodology, computational simulations of some optimized microchannel layouts are employed to validate the implemented topology optimization algorithm. Some obtained results are shown to illustrate the methodology.

1 INTRODUCTION

In fluid flow systems, one important matter is power dissipation along channels which leads to a pressure drop, compromising their correct operation and their efficiency. This demands a detailed analysis of power consumption processes to achieve the most efficient system design, allowing smaller power consumption, and consequently lower costs and a lower environment impact, which is also a great concern recently.

The application of channels in fluid flow has been occurred intensively in many engineering areas, from transportation (large scale) to biological fluid conduction (small scale). As a practical application we can mention microchannel heat sink, which has been studied for a long time as alternative to increase cooling efficiency of small and powerful electronic devices. This technology was introduced by Tuckerman and Pease (1981) who performed experiments on silicon based microchannel heat sink for electronic cooling. Microchannels constructed on a conductivity body can provide large convective heat transfer coefficient, and small mass and volume for heat sink designs, which makes it very suitable for cooling compact electronics devices such as high-end microprocessors applied to general computation. The microprocessors dissipate a large amount of heat, and need very efficient dissipation system to avoid malfunction or even product damage. As long as these microprocessors become more powerful and smaller, the need of efficiency in heat dissipation is even more evident.

To achieve a better microchannel heat sink performance, it is crucial that these cooling systems have a very efficient fluid flow channel, minimizing pressure drops along its extension, and consequently allowing an efficient thermal dissipation process. Through the last three decades, many studies have been conducted, in order to achieve fluid flow microchannels with better configuration for minimizing power dissipation. Several analytical, numerical, and experimental studies have carried out the design optimization of microchannel heat sinks to determine the geometric dimensions that give the best performance (Knight et al., 1992; Toh et al., 2002; Qu and Mudawar, 2002).

Some analytical studies have employed the classical fin theory that models the solid walls, which separate the microchannels, as the thin fins (Knight et al., 1992). Nevertheless, most microchannel design analyses in the literature are carried out through numerical studies, in which the microchannel domains are simply modeled by using the conventional fluid flow and heat transfer governing equations (Toh et al., 2002). In these studies, parametric optimization problems are solved by computation models, which domain consists of only a single channel and the corresponding slice of wall with symmetrical boundary conditions. Although extract experimental data in microchannel devices by using conventional measurements techniques is often difficult, in general experimental studies are employed to validate computational models (Qu and Mudawar, 2002).

Optimization methods with numerical analyses offer advantages to analyze the microchannel behavior, allowing the study of more complex cases with multi-objective applications. The basis of numerical studies on channel flow optimization was given by Pironneau (1973), who conducted a shape optimization analysis in airfoils and other devices, such as diffusers. In his studies, the shape optimization process has been applied to obtain minimum drag profiles and minimum pressure drop diffusers.

The main objective of this work is to present a methodology for obtaining optimal microchannel layouts applied to a heat sink design using Topology Optimization Method (TOM). Later in the 90’s, there has been a great development of the TOM, which essentially distributes limited amount of material inside a design domain to optimize a cost function requirement, satisfying some specified constraints (Bendsøe and Sigmund, 2003). Nowadays, TOM has been employed to several applications, such as fluid flow problems (Borrvall and Petersson, 2003). In this case, instead of controlling only solid material and void regions, as often performed for structural analysis field (Bendsøe and Kikuchi, 1988), the interest is focused on distributing liquid and solid materials, and then, creating optimized fluid flow systems that minimizes the total power of the Stokes flows. Recently, other works that extend the application of TOM to this field have been conducted. For instance, Gersborg-Hansen et al. (2005) consider the problem of optimal design of flow domains for Navier–Stokes flows. Guest and Prévost (2006) presented a material model in which the design domain for topology optimization is treated as a porous medium with flow governed by Darcy’s law. Aage et al. (2008) extend the Stokes flow applications to large scale 2D and 3D problems.

One of the great advantages of the TOM is the possibility of analyzing a much wider range of solutions, due to the “free” material distribution method. By applying this method, an optimal solution can be achieved with no need of proposing a “pre-structured” initial guess, which tends to limit the final solution, by directing the optimization process. This issue may be a problem in parametric and shape optimization, where a preliminary solution model must be stated. Thus, “not-so-intuitive” solution can be recovered, as shown in Borrvall and Petersson (2003). This work aims to find “not-so-intuitive” optimal solution for channel layouts to attend a multi-objective function: minimize the pressure drop of fluid flow, and maximizing the heat dissipation along the channel, mainly focused for microchannel heat sink design.

The methodology proposed in this work is presented in the next sections. Section 2 describes the fundamental theory. Section 3 presents the finite element modeling. Section 4 shows an overview of the topology optimization procedure. Section 5 details the results obtained at this moment. Finally, in Section 6 some discussion about obtained results and conclusion are given.

2 FUNDAMENTAL THEORY

The fundamental theory is given by the constitutive equations for Newtonian fluid flow, based on the well-known Navier-Stokes equations, given by:

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

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

where ρ_m is the fluid mass density, μ is the dynamic viscosity, $\mathbf{u}$ is the velocity field, p is the pressure, and $\mathbf{f}$ is the body load. Equation (1) refers to the conservation of momentum, and Eq. (2) refers to conservation of mass, or continuity equation.

As in Borrvall and Petersson (2003), in this study the Navier-Stokes equations are simplified to a linear form, considering a steady-state, incompressible fluid flow at low Reynolds, where the viscous effects overlap the inertia effects. Thus, the following Stokes flow equations are obtained:

$$-\mu \nabla^2 \mathbf{u} + \nabla p = \mathbf{f} \tag{3}$$

$$\nabla \cdot \mathbf{u} = 0 \tag{4}$$

Equation (3) dictates the fluid flow, and it is coupled with Eq. (4), which acts like a constraint in the velocity field to ensure the incompressibility condition.

Here, the material model proposed by Guest and Prévost (2006) is adopted to describe the behavior of the solid region and fluid flow in free regions of the domain. They studied a formulation that mixes a fluid-like and a solid-like behavior by combining the standard Stokes flow equation with an equation that describes a porous medium flow, known as Darcy flow equation, given by:

$$\begin{aligned} \alpha\mathbf{u} &= \left(\nabla p - \mathbf{f}\right) \\ \nabla\cdot\mathbf{u} &= 0 \end{aligned} \tag{5}$$

where α is the inverse permeability of the porous medium region.

The main idea is to apply the Stokes equations to model the fluid flow behavior, and to control the velocity field in solid regions through the Darcy's law, by assuming it to be a porous medium with nearly-zero flow permeability.

A material model is necessary to describe continuity of the fluid velocity and pressure fields across the solid interface. Thus, equations (3) and (5) can be combined through the Brinkman's equation, given by:

$$\begin{aligned} \mu\nabla^2\mathbf{u} + \alpha\mathbf{u} &= \nabla p - \mathbf{f} \\ \nabla\cdot\mathbf{u} &= 0 \end{aligned} \tag{6}$$

Equation (6) has a penalization term controlled by α, which is applied to force a very small flow in solid regions. This approach allows the optimization to work with a continuum variation on the liquid-solid model, which is relevant during the optimization process.

Additionally to the fluid movement equations, following convection-diffusion heat transfer governing equation, in steady-state flow, is considered to model the heat dissipation of the domain:

$$\rho_m(c_p\mathbf{u})\cdot\nabla T + f_T = k\nabla^2 T \tag{7}$$

where T is the temperature, f_T is the heat generation, ρ_m, c_p, and k are fluid density, specific heat, and thermal conductivity, respectively.

3 FINITE ELEMENT MODELING

The finite element method (FEM) is applied to solve the equations (6) and (7), described in previous section. For fluid flow problem, the design domain is divided by using bilinear rectangular elements, which have four nodes for velocity field (two degrees of freedom per node) and one node for pressure field. Although it is recognized as a not full-stable element according to the LBB or div-stability-condition (Hughes et al., 1986), the adopted element has shown a good accuracy for velocity field calculation, with expected spurious oscillation in the pressure field caused by the velocity and pressure fields coupling, which in this particular application do not affect the results decisively (Borrvall and Petersson, 2003).

By applying the FEM to Eq. (6), and writing it to the discrete matrix form, the following equation system is obtained:

$$\begin{bmatrix} \mathbf{K} & -\mathbf{G}^T \\ \mathbf{-G} & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \mathbf{p} \end{Bmatrix} = \begin{Bmatrix} \mathbf{f} \\ \mathbf{0} \end{Bmatrix} \tag{8}$$

where **u** is the nodal velocity, **p** is the nodal pressure, and **f** is the nodal body load. **K** is the velocity stiffness matrix, **G** denotes the divergent operator matrix for the continuity equation, and $\mathbf{G}^T$ the gradient operator matrix. Here, **K** and **G** are given by (Borrvall and Petersson, 2003):

$$\mathbf{K} = \int_V \left(\mu \mathbf{B}^T \mathbf{I_0} \mathbf{B}\right) dV + \int_V \left(\alpha(\rho) \mathbf{N}^T \mathbf{N}\right) dV$$
$$\mathbf{G} = \int_V \left(\left(\nabla \mathbf{N_u}\right)^T \mathbf{N_p}\right) dV \qquad (9)$$

with

$$\mathbf{I_0} = \begin{bmatrix} 2 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \qquad (10)$$

where **B** and **N** are the well known strain–displacement matrix and standard shape function matrix for velocity field (Zienkiewicz e Taylor, 2002), respectively. $\alpha(\rho)$ is the inverse permeability function with parameter ρ, which determines the material at each point of the porous medium (Borrvall and Petersson, 2003).

Now, considering the heat transfer problem, the design domain is also divided by using rectangular elements, however using four nodes for temperature field (one degree of freedom per node). Thus, by applying the FEM with Streamline-Upwind Petrov-Galerkin formulation to Eq. (7), which avoids spurious instabilities in the convection-diffusion discretized problem (Brooks e Hughes, 1982), the following matrix form equation system is obtained:

$$\left(\mathbf{n}_T + \mathbf{k}\right)\mathbf{T} = \mathbf{f}_T \qquad (11)$$

with

$$\mathbf{n}_T = \int_V (\mathbf{w}_{PG})^T \rho_m c_p (\mathbf{w}_G \mathbf{u})^T (\nabla \mathbf{w}_G) dV$$
$$\mathbf{k} = \int_V (\nabla \mathbf{w}_G)^T k (\nabla \mathbf{w}_G) dV \qquad (12)$$
$$\mathbf{f}_T = \int_V (\mathbf{w}_G)^T Q dV$$

where $\mathbf{w}_G$ and $\mathbf{w}_{PG}$ are weighting functions (Zienkiewicz e Taylor, 2002).

By solving the system presented in Eq. (8), considering a correct set of boundary conditions, the velocity and pressure fields can be determined. After that, the temperature field is obtained through Eq. (11).

4 TOPOLOGY OPTIMIZATION PROBLEM

4.1 Topology optimization concept

In the topology optimization, the design domain is discretized by using a FEM mesh, and in this work, each element of a fixed design domain can assume either fluid or solid material, according to the material model.

The material model combines the characteristics of both materials (solid and fluid), defining the material for each element of the discretized domain. It is controlled by the design variable ρ, in such way that for ρ=0 one retains a solid material, and for ρ=1 one retains a fluid material, characterizing a discrete 0-1 problem. Intermediate values of ρ do not have physical application and is not desirable at the final design. However, it is very common to work with a continuous problem, allowing ρ to assume intermediate values for preventing well-known solution problems in the discrete model (Bendsøe and Sigmund, 2003).

Here, the material model is described by Eq. (6), in which the inverse permeability (α) is a continuous function of the design variable ρ (Borrvall and Petersson, 2003; Gersborg-Hansen et al., 2005). This model essentially describes a Stokes flow behavior for fluid elements. For solid elements, the combined porous medium model predominates, with permeability controlled by the design variable ρ, such that for a full solid element ($\rho \to 1$) the velocity is close to zero ($u \to 0$).

4.2 Formulation of the topology optimization problem

The total potential power evaluated at the solution obtained by the FEM analysis is adopted as objective function for the fluid flow problem. Considering a common case, where there are no body forces over the fluid domain, the total potential power represents the power dissipation on the fluid in the design domain, given by:

$$\Phi = \mathbf{u}^T \mathbf{K}\,\mathbf{u} \tag{13}$$

where **u** represents the nodal velocity field vector and **K** is the velocity stiffness matrix.

Equation (13) has the same discrete form of the well-known mean compliance used very often in structural optimization (Bendsøe and Sigmund, 2003), and may represent the mean pressure drop over the channel. The goal is to minimize power dissipation, and consequently to minimize the pressure drop.

To maximize heat dissipation, another objective function must be stated. The chosen cost function utilizes the temperature distribution, obtained from the Eq. (7), to evaluate the system heat transfer performance. It has also the same discrete form as the mean compliance problem, as follows:

$$\Gamma = \mathbf{f}_T{}^T \mathbf{T} \tag{14}$$

where **T** is nodal temperature field vector and $\mathbf{f}_T$ represents the heat generation.

These two goals (to minimize the pressure drop and to maximize heat transfer) are evaluated through a multi-objective function which allows the design process to give priority to one of them, or treat both equally, as follows:

$$\Psi = u \ln(\Phi) + w \ln(\Gamma) \tag{15}$$

where u and w are weighting factors.

Thus, the topology optimization problem is stated for the channel optimization, in a discrete form, as:

$$\begin{aligned} \underset{\boldsymbol{\rho}}{\text{minimize}}: \quad & \Psi = u \ln(\Phi) + w \ln(\Gamma) \\ \text{Subject to:} \quad & \begin{bmatrix} \mathbf{K} & -\mathbf{G}^T \\ \textbf{-G} & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \mathbf{p} \end{Bmatrix} = \begin{Bmatrix} \mathbf{f} \\ \mathbf{0} \end{Bmatrix} \\ & (\mathbf{n}_T + \mathbf{k})\mathbf{T} = \mathbf{f}_T \\ & \sum_{i=1}^{N} \rho_{\mathrm{i}} \le V; \quad 0 \le \rho_{\mathrm{i}} \le 1 \end{aligned} \tag{16}$$

A volume fraction constraint, denoted by the ratio of fluid material volume over the total domain volume, is adopted in this topology optimization problem.

4.3 Topology optimization precedure

In this work, the Topology Optimization Method (TOM) combines the Sequential Linear Programming (SLP) (Haftka, 1996) with FEM to solve the topology optimization problem. The procedure of the TOM begins with the initial domain and boundary conditions definition. The design domain is discretized conveniently, so the FEM analysis can be performed and the optimization process follows. The topology optimization algorithm calculates the cost function value at each iteration and performs a sensitivity analysis of this cost function based on gradient calculations over the design variables. The optimization algorithm uses this sensitivity analysis as guidance for recalculating material distribution along the design domain, and updates the information, until an optimized topology is obtained. The typical TOM procedure adopted in this work is shown in Figure 1.

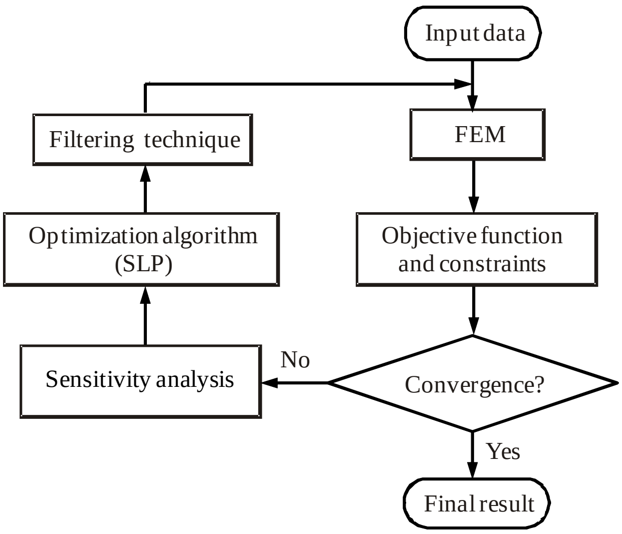


Figure 1: Flow chart of topology optimization algorithm.

The sensitivity analysis is performed by calculating the gradients of functions Φ and Γ in the multi-objective function of Eq. (15) in relation to the design variables (ρ), using adjoint method (Bendsøe and Sigmund, 2003).

The projection scheme proposed by Guest et al. (2004) is employed in this work as filtering technique, to avoid mesh-dependency and checkerboard problems, often found in topology optimization (Bendsøe and Sigmund, 2003).

5 RESULTS

This section shows the preliminary results obtained. It will be illustrated initially, minimum pressure drop cases, and then some results combined with heat transfer process.

5.1 Minimum pressure drop channel design

The first example studied is a minimum pressure drop case. In this example, internal fixed solid regions are considered in the design domain, as shown in Figure 2. This model has one inlet and one outlet region, both set to a unitary velocity. All other domain boundaries are prescribed as walls, with non-slip and impermeability conditions. At the outlet region, the pressure is set to 0, acting like a sink.

The results presented in Figure 3 are obtained for a domain composed by 80x80 elements. White regions represent fluid material and black regions represent solid regions. Figure 3d shows the fluid flow velocity distribution in the domain, considering volume fraction constraint V=0.4. This test shows how the optimization problem is stated and illustrates also the volume fraction influence over the solution.

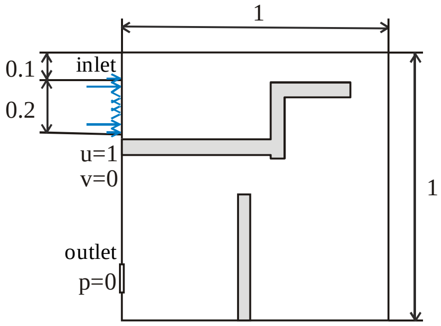
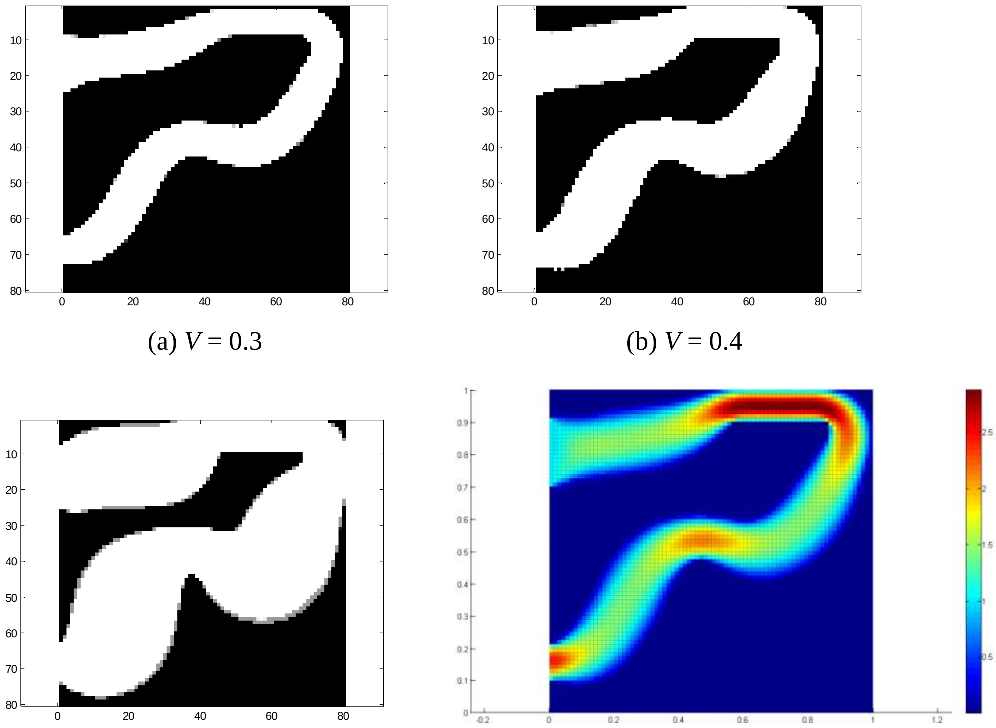


Figure 2: Domain with internal solid regions (obstacles).



(a) V = 0.3

(b) V = 0.4

(c) V = 0.6

(d) velocity field (V = 0.4)

Figure 3: Obtained results considering different volume fraction constraints (V).

5.2 Heat sink design

In this example, the topology optimization algorithm of this work is applied to a heat sink design. Essentially, optimization process is carried out to achieve a channel with both low pressure drop and high heat dissipation attributes, increasing a heat sink device.

This heat sink design domain has one inlet and two outlet regions, with direction defined as shown in Figure 4. The boundary conditions, as well as the symmetric model used to save computational time, are also shown in Figure 4. At the inlet region, parabolic flow velocity profile (v_{max}=1) and fixed temperature of 20 °C are prescribed. The pressure at outlet regions is set to zero, acting similar to a sink. A uniform heat source is distributed along the whole domain. All external walls of the domain are prescribed as non-slip adiabatic boundaries. For this example, thermodynamic constants of the fluid (water) and the solid material (Aluminum) are considered.

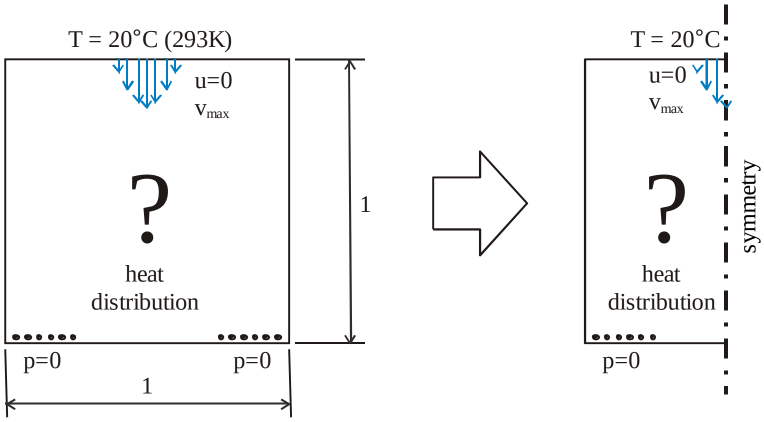
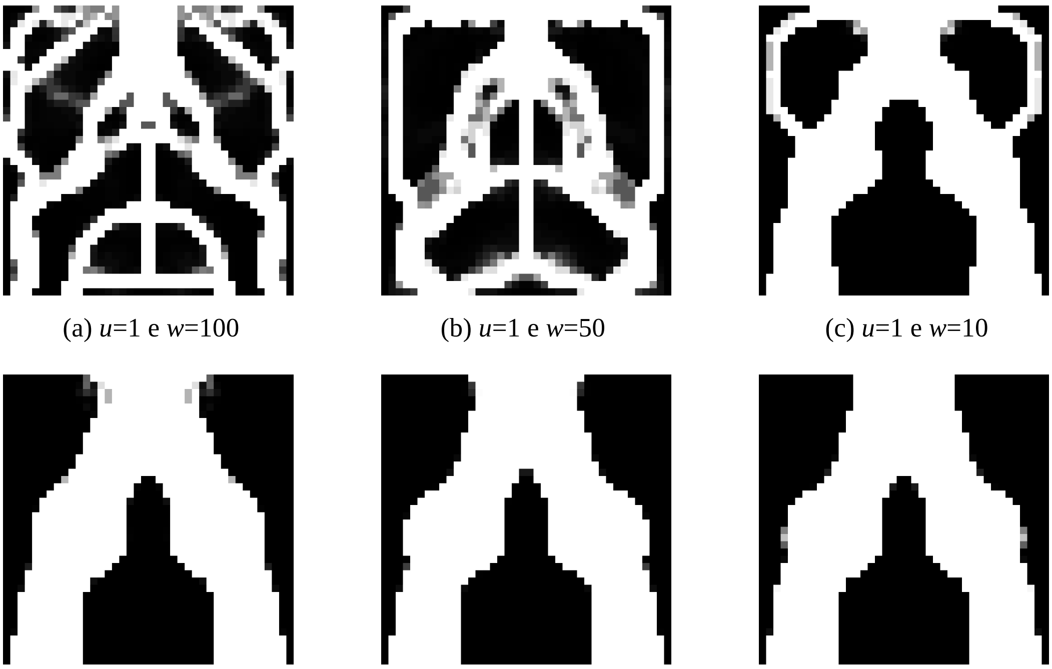


Figure 4: Heat sink design domain

The results presented in Figure 5 are obtained for the symmetric domain discretized by 40x20 elements. A volume fraction constraint equal to 0.4 is adopted for all examples. These results show how the optimization problem is stated and illustrate also the weighting factors (*u* and *w*) influence over the solution.

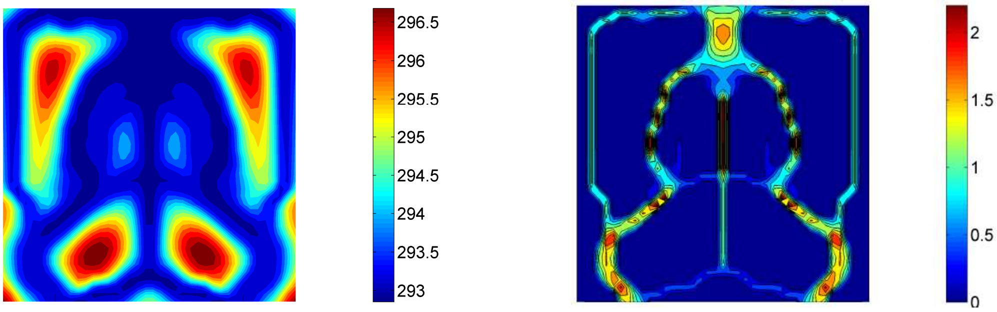


(a) *u*=1 e *w*=100 (b) *u*=1 e *w*=50 (c) *u*=1 e *w*=10

(d) *u*=1 e *w*=1 (e) *u*=10 e *w*=1 (f) *u*=100 e *w*=1

Figure 5: (a) Φ=0.346 and Γ=2.545x10^7; (b) Φ=0.228 and Γ=2.552x10^7; (c) Φ=0.014; Γ=2.607x10^7; (d) Φ=8.642x10^{-3} and Γ=2,624x10^7; (e) Φ=7.375x10^{-3} and Γ=2.628x10^7; (f) Φ=6.853x10^{-3} and Γ=2.752x10^7.

As expected, as *w* increases the thermal dissipation (Γ) decreases. On the other hand, as *u* increases the pressure drop (Φ) minimization is evident. Figure 6 shows the temperature distribution and the heat flux for the obtained channel configuration shown in Figure 5b.



(a) (b)

Figure 6: (a) Temperature and (b) flow velocity field for configuration shown in the Figure 5b.

6 DISCUSSION AND CONCLUSION

The application of the topology optimization in fluid mechanics and heat transfer is practical and it allows the systematic design of fluid flow channels. This approach could be used for more efficient microchannel heat sink design, which has a lot of possible applications.

Volume fraction verification has been performed and it is possible to visualize its influence over the results. According some tests, it is also verified that the implemented topology optimization algorithm is mesh-independent. An example of application of the combined fluid flow and heat transfer characteristics for channel design is shown. This example illustrates the viability of applying the topology optimization process to achieve a channel design combining these two distinct characteristics (fluid flow and heat transfer) at the same time.

The application of combined Stokes-Darcy flow equation has been shown very efficient within the topology optimization algorithm. Although it has certain limitations, such as it is a linear approach of the full Navier-Stokes equations and suitable only for low Reynolds fluid flow, this model is applicable for many different problems, from bend-pipes and diffusers to more complex cases.

Weighting factors (*u* and *w*) which allows to control tuning for fluid flow or heat transfer behavior has been performed and it is noticed its influence over the results. As there is a tradeoff between fluid flow layout that maximize the heat transfer, the algorithm tries to increasing the heat exchanging area by introducing some small channels around the principal (major) channel, as can be seen in Figure 5.

As conclusion, authors consider the application of the topology optimization in fluid flow channel design a very promising field. The analysis reported here can be extended for many other study cases, which utilize cooling devices design based on fluid flow channel. Thus, experimental characterization and manufacturing prototypes of micro scale heat sink configurations will be done as next step of this work.

ACKNOWLEDGEMENTS

The authors would like to acknowledge FAPESP, CAPES, and CNPq for the financial support provided for this project through fellowships and CNPq research projects (472141/2008-2 and 500991/2009-0).

REFERENCES

Aage, N., Poulsen, T. H., and Gersborg-Hansen, A., Sigmund, O., Topology optimization of large scale stokes flow problems, *Structural And Multidisciplinary Optimization*, 35: 175-180, 2008.

Bendsøe, M. P., and Kikuchi, N., Generating optimal topologies in structural design using homogenization method, *Comput. Methods Appl. Mec. Eng.*, 71: 197-224, 1988.

Bendsøe, M. P., and Sigmund, O., *Topology Optimization: Theory, Methods and Applications*, Springer-Verlag, Berlin, 2003.

Borrvall, T., and Petersson, J., Topology optimization of fluids in Stokes flow, *International Journal for Numerical Methods in Fluids*, 41: 77–107, 2003.

Brooks, A.N., and Hughes, T. J. R., Streamline Upwind Petrov-Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier-Stokes Equations, *Computer Methods In Applied Mechanics And Engineering*, 32: 199-259, 1982.

Gersborg-Hansen, A., Sigmund, O., and Haber, R. B., Topology optimization of channel flow problems, *Structural And Multidisciplinary Optimization*, 30: 181-192, 2005.

Guest, J. K., and Prévost, J. H., Topology optimization of creeping fluid flows using a Darcy-Stokes finite element, *International Journal For Numerical Methods In Engineering*, 66: 461-484, 2006.

Guest, J. K., Prévost, J. H., and Belytschko, T., Achieving minimum length scale in topology optimization using nodal design variables and projection functions, *International Journal For Numerical Methods In Engineering*, 61 (2): 238–254, 2004.

Haftka, R. T., and Gandhi, R. V., Structural Shape Optimization – A survey, *Computer Methods in Applied Mechanics and Engineering*, 57: 91-106, 1986.

Hughes, T. J. R., Franca, L.P., and Balestra, M., A new finite element formulation for computational fluid dynamics, V. circumventing the Babuska-Brezzi condition, A stable Petrov-Galerkin formulation of the Stokes problem accommodating equal-order interpolations, *Computer Methods In Applied Mechanics And Engineering*, 59: 85-99, 1986.

Knight, R.W., Hall, D.J., Goodling, J.S., and Jaeger, R.C., Heat sink optimization with application to microchannels, *IEEE Transactions on Components, Hybrids, and Manufacturing Technology* 15: 832–842, 1992.

Pironneau, O., On optimal profiles in Stokes flow, *Journal of FluidMechanics*, 59: 117–128, 1973.

Qu, W., and Mudawar, I., Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink, *Int. J. Heat Mass Transfer*, 45: 2549–2565, 2002.

Toh, K.C., Chen, X.Y., and Chai, J.C., Numerical computationof fluid flow and heat transfer in microchannels, *International Journal of Heat and Mass Transfer*, 45: 5133–5141, 2002.

Tuckerman, D. B., and Pease, R. F. W., High-performance heat sinking for VLSI, *IEEE Electron Device Lett.*, EDL-2: 126–129, 1981.

Zienkiewicz, O. C., and Taylor, R. L., *The Finite Element Method*, 5th Ed, volume III. Oxford: Butterworth-Heinemann, 2002.