Topology Optimization of Thermal Heat Sinks

J. H. K. Haertel[1]*, K. Engelbrecht[1], B. S. Lazarov[2] and O. Sigmund[2]
[1] Department of Energy Conversion and Storage, Technical University of Denmark
[2] Department of Mechanical Engineering, Technical University of Denmark
*Corresponding author: Frederiksborgvej 399, Building 779, 4000 Roskilde, Denmark, jhkh@dtu.dk

Abstract: In this paper, topology optimization is applied to optimize the cooling performance of thermal heat sinks. The coupled two-dimensional thermofluid model of a heat sink cooled with forced convection and a density-based topology optimization including density filtering and projection are implemented in COMSOL Multiphysics. The optimization objective is to minimize the heat sink's temperature for a prescribed pressure drop and fixed heat generation. To conduct the optimization, COMSOL's Optimization Module with GCMMA as the optimization method is used. The implementation of this topology optimization approach in COMSOL Multiphysics is described in this paper and results for optimized two-dimensional heat sinks are presented. Furthermore, parameter studies regarding the effect of the prescribed pressure drop of the system on Reynolds number and realized heat sink temperature are presented and discussed.

Keywords: Topology optimization, thermofluid modeling, heat sink, forced convection, PDE filter.

1. Introduction

Topology optimization is becoming increasingly popular as a design method for multiphysics systems in general and thermofluid systems in particular [1,2]. Traditional designs are based either on engineering intuition or on trial and error approaches. In contrast, topology optimization [3] provides a systematic way to account for the complex environment in the design process and therefore can yield unintuitive designs.

The focus of the work is to describe the implementation of the three-field density-based topology optimization model [4] in COMSOL Multiphysics. The three-field model resembles many micro- and nano- scale production processes [5,6] and provides a base for considering manufacturing uncertainties [4] in the optimization.

In density-based topology optimization, the algorithm determines the optimal distribution of a material in a given design domain. In the specific case considered in this paper, solid heat sink material is distributed in order to minimize a selected objective. The rest of the domain is occupied with a cooling fluid. The material distribution is modeled with the help of a density field which takes the value 1 if a point is occupied with fluid and 0 if a point is occupied with solid. In order to utilize gradient-based optimization methods, the material distribution problem is relaxed to take values between zero and one.

Topology optimization of thermofluid systems has been presented for example in [7,8,9,10] for forced convective heat transfer, and in [11] for natural convection problems. In this paper, the topology optimization of heat sinks cooled with forced convection is presented. The heat sink is used to cool a surface with a constant heat production and the pressure drop in the fluid is constrained. The optimization objective is the minimization of the solid plate temperature of the heat sink. This objective combined with the pressure drop constraint yields an interesting optimization problem from an engineering point of view.

The thermofluid modeling is described in section 2 and a brief introduction to the applied topology optimization method including density filtering and projection is given is section 3. Section 4 is concerned with the model and optimization implementation in COMSOL Multiphysics. The optimization results are presented in section 5. Optimized heat sink structures and the corresponding temperature and fluid velocity distributions are depicted as well as parameter studies regarding the cooling effect and the system's Reynolds number dependency of the prescribed pressure drop.

2. Thermofluid Modeling and Governing Equations

In the following sections, the thermofluid modeling of the system is presented. The basic

idea of the 2D modeling of the forced convection heat sink is shown in (Figure 1). Details about the COMSOL Multiphysics implementation of the model are described in section 4.
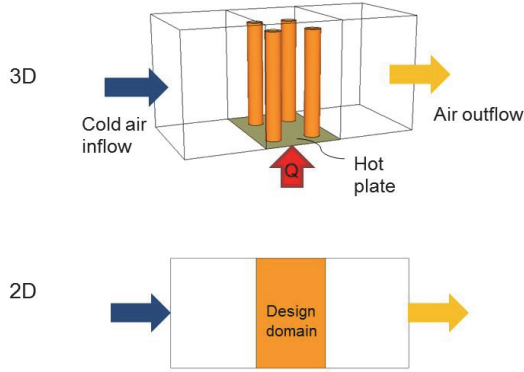

Figure 1. Sketch of the modeled forced convection heat sink in 3D and 2D.

2.1 Fluid Dynamics Modeling

The fluid dynamics are modeled under the assumption of a stationary laminar flow with an incompressible fluid. Furthermore, a 2D fluid flow is assumed which can be motivated by stating that the heat sink fins are much longer than the solid base plate. This leads to the 2D Navier-Stokes equation (1) and continuity equation (2).

$$\rho_{fl} \cdot (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = -\nabla p + \mu(\nabla^2 \boldsymbol{u}) + \boldsymbol{F} \qquad (1)$$

$$\rho_{fl}(\nabla \cdot \boldsymbol{u}) = 0 \qquad (2)$$

In the Navier stokes equation a Brinkman friction term (equation (3)) is introduced to penalize fluid velocities inside the solid material in the design domain. Outside the design domain this term is omitted so that the Navier-Stokes equation describes normal fluid flow.

$$\boldsymbol{F}(\gamma) = -\overline{\alpha} \frac{1 - \gamma}{1 + b_\alpha \gamma} \boldsymbol{u} \qquad (3)$$

Where γ is the design variable and the used interpolation is given in [11] which is a slightly modified version of the original interpolation used in [12]. The parameter b_α determines the convexity of the interpolation. In all

interpolations used in this work γ=0 corresponds to solid and γ=1 corresponds to fluid.

2.2 Heat Transfer Modeling

The heat transfer in the fluid outside the design domain is modeled according to equation (4). Equation (5) describes the modeling of the heat transfer within the design space which includes an interpolation of the thermal conductivity based on γ. The conductive heat transfer in the solid plate is stated in equation (6).

$$\rho_{fl}c_{fl}\boldsymbol{u} \cdot \nabla T_{fl} - \nabla \cdot (k_{fl}\nabla T_{fl}) = 0 \qquad (4)$$

$$\rho_{fl}c_{fl}\boldsymbol{u} \cdot \nabla T_{fl} - \nabla \cdot (k_{fl}I_k(\gamma)\nabla T_{fl})$$
$$= \frac{q_{conv}(\gamma)}{dz_{fl}} \qquad (5)$$

$$\nabla \cdot (k_s\nabla T_s) + \frac{q_{prod}}{dz_s} - \frac{q_{conv}(\gamma)}{dz_s} = 0 \qquad (6)$$

Where the parameters dz_{fl} and dz_s describe the respective thickness of the fluid layer and the solid plate.

The heat transfer between the design domain layer and the solid plate base layer is modeled by a heat flow having the mathematical structure of convective heat transport with a variable heat transfer coefficient. This allows to model high heat flow into the design domains in case of solid, low heat flow in the case of fluid, and interpolations in between. The case of solid represents high conductive heat flow from the base plate layer into the heat sink's fins and the low heat flow for fluid represents a purely convective heat transfer from the solid base layer into the fluid layer. A similar idea of a thermal base layer and a fluid-thermal design layer with heat transfer in between, however implemented in a different way than in this work, was presented in [13]. The heat transfer term utilized in this model is defined as follows:

$$q_{conv}(\gamma) = h_{min}I_h(\gamma)\,(T_s - T_{fl}) \qquad (7)$$

The interpolation function for the thermal conductivity and the heat transfer is a RAMP-style function as used in [11] and originally presented in [14]. The formula is stated in equation (8).

$$I_i(\gamma) = \frac{\gamma(C_i(1+b_i)-1)+1}{C_i(1+b_i\gamma)} \qquad (8)$$

$$C_k = \frac{k_{fl}}{k_s} \qquad (9)$$

$$C_h = \frac{h_{min}}{h_{max}} \qquad (10)$$

For both thermal conductivity and heat transfer interpolations respective parameters b_k and b_h determine the convexity of the interpolation function in equation (8). The parameter C_i in equation (8) is defined by equation (9) for the conductivity interpolation and by equation (10) for the heat transfer interpolation.

3. Topology Optimization

3.1 Problem Formulation

The considered optimization objective is to minimize the average temperature of the solid plate with heat production which depends implicitly on the design variable field γ and the system's state variables s. Thus, the topology optimization problem can be defined in the following way:

$$
\begin{aligned}
\text{min.:} \quad & T_{s,average}(\boldsymbol{\gamma}, \boldsymbol{s}) \\
\text{s.t.:} \quad & \boldsymbol{r}(\boldsymbol{\gamma}, \boldsymbol{s}) = \boldsymbol{0} \qquad (11)\\
& 0 \le \gamma_i \le 1 \quad \text{i=1,…,n}
\end{aligned}
$$

Where **r(y,s)** is the residual of the discretized system of state equations that are described in section 2. Apart from the indicated constraints, no additional constraints need to be imposed since system constraints such as the prescribed pressure drop are already considered in the thermofluid model.

3.2 Filter and Projection

A density filter should be used in fluid-thermal topology optimization to avoid problems with ill-posedness of the optimization problem and to introduce a minimum length-scale into the design. In this work, a Helmholtz-type PDE filter [15] is used since it allows for a computationally efficient density filtering and can be easily implemented in COMSOL Multiphysics. The filter PDE is given by

$$-r_f{}^2 \nabla^2 \tilde{\gamma} + \tilde{\gamma} = \gamma \qquad (12)$$

where r_f is the filter parameter. A drawback of density filtering is the inherent introduction of a band with intermediate densities between the solid and fluid regions. This band can be reduced by projecting the filtered design variable field towards 0 and 1, thus obtaining a design with sharper transitions from fluid to solid. For this purpose, a smoothed Heaviside projection [4] is used which is defined as:

$$\bar{\tilde{\gamma}}_i = \frac{\tanh(\beta\eta) + \tanh(\beta(\tilde{\gamma}_i - \eta))}{\tanh(\beta\eta) + \tanh(\beta(1-\eta))} \qquad (13)$$

Where η is a threshold parameter and β determines the steepness of the projection. It should be stated that the projected $\bar{\tilde{\gamma}}$ field becomes the physically meaningful one that is used in the interpolation functions which are presented in chapter 2.

4. Use of COMSOL Multiphysics

Within the presented studies, air is used as fluid and copper as heat sink material. COMSOL's Material Library is used to define the material properties. A symmetry condition is imposed along the axial center line so that the problem size is halved. The basic dimensions of the topology optimization model are indicated in Table 1.

Table 1: Basic dimensions of the topology optimization model.

Length channel	5.5 mm
Length design domain and solid plate	1.5 mm
Width channel and solid plate	4 mm
Height channel	10 mm

4.1 Multiphysics Implementation

Systems with Reynolds numbers up to 150 are considered in the analysis, thus the fluid flow is laminar. Therefore, COMSOL's Laminar Flow physics interface is used to solve the fluid mechanical system stated in equation (1) and

equation (2). The pressure difference between inlet and outlet is fixed and a no slip condition is imposed at the channel side. A first order discretization is used for the velocity and pressure field and pseudo-time stepping is enabled to increase the robustness of the solver.

The heat transport problem which is stated by equation (4), (5), and (6) is implemented in the Heat Transfer Module. The fluid's temperature at the inlet is fixed to 290 K and a total heat production of either 0.025 W or 0.05 Watt is set in the solid plate. Thermal insulation is modeled on the channel wall. For both the fluid and the solid temperature field, a linear discretization is used. To reduce the computational complexity, the fluid mechanics solution was decoupled from the heat transfer equations. Therefore, only the heat transfer solution depends on the fluid velocity field but the velocity field is determined under the assumption of a constant fluid temperature. This assumption should be a legitimate approximation since the changes of the fluid temperature are smaller than 20 K in the analyzed system.

4.2 Implementation of Topology Optimization

The topology optimization is implemented within the Optimization Module using an element-wise constant design variable field. The interpolation functions for the friction force, thermal conductivity, and out-of-plane heat transfer being stated in equation (3) and (8) are implemented in COMSOL as variables depending on the design variable field.

COMSOL's Coefficient Form PDE interface is used to implement the PDE filter described in equation (12) and a linear discretization is chosen for the filtered design variable field. The threshold parameter is set as 0.5 and the filter parameter is set as 1.5 times the minimum mesh size to avoid numerical instabilities which can occur with smaller filter radii. Zero flux boundaries are set for the filter PDE at the symmetry line and the outside border of the channel. At the remaining boundaries, a design variable value of one which corresponds to pure fluid is enforced which prevents solid from being "glued" to the design domain boundaries. The projection of the filtered field stated in equation (13) is realized with an analytical function in COMSOL.

The optimization is conducted using the GCMMA optimization method [16]. During the optimization, the convexity parameters of the interpolation functions and the steepness parameter of the projection are ramped as needed. This continuation approach [17] is conducted to obtain a more convex optimization problem at the beginning and to subsequently increase the penalization of intermediate densities during the optimization. COMSOL's LiveLink for MATLAB is used to automate the parameter ramping within a MATLAB script.

5. Results and Discussion

The design variable field of the optimized heat sink is shown in (Figure 2) for a prescribed pressure drop of 0.75 Pa and a total heat production of 0.05 W in the solid plate (corresponding to a heat flux of 8333 W/m^2). It can be seen that the topology optimization generated a structure with four fins and three fluid channels. Moreover, it should be noted that the fins have small bumps which presumably enhance the heat transfer into the fluid. The corresponding velocity magnitude distribution and streamlines of the velocity field are depicted in (Figure 3). (Figure 4) shows the corresponding temperature distribution in the fluid design layer.
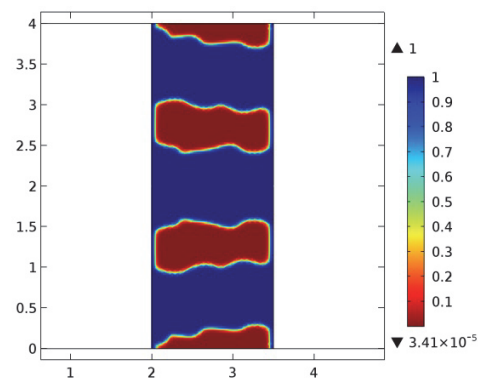

Figure 2. Design variable field of optimized heat sink. A density of 0 indicates solid material and 1 indicates flow passages.
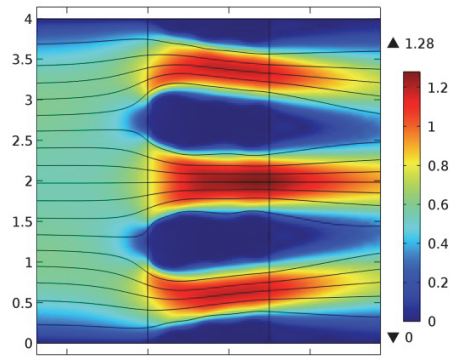
Figure 3. Fluid velocity magnitude (m/s) and streamlines for optimized heat sink.
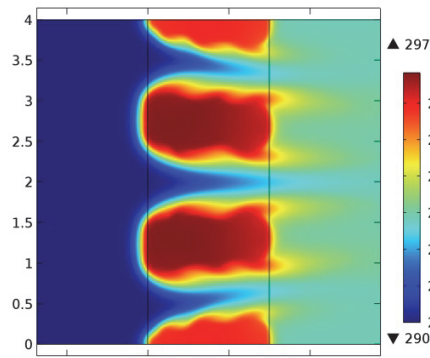

Figure 4. Temperature distribution (K) for optimized heat sink.

A parameter study showing the optimized heat sink's solid plate temperature plotted against the prescribed system pressure drop is shown in (Figure 5). Two heat production rates of 0.025 W and 0.05 W are included in the study. For both heat production rates, the solid plate's temperature decreases monotonously with increasing pressure drop. (Figure 6) depicts the dependency of the optimized system's Reynolds number of the prescribed pressure drop. The channel width was used as the characteristic length for the Reynolds number, and the average velocity at the inlet was used as the characteristic velocity. The Reynolds number increases with increasing pressure drop and maximum Reynolds numbers of around 140 are reached. The Reynolds number decrease between the pressure drop of 1 Pa and 1.25 Pa for a heat production of 0.025 W and between the pressure drop of 0.75 Pa and 1 Pa for a heat production of 0.05 W is explained by the addition of a new fin

to the system. Thereby, the cooling performance is increased but the system operates with a lower volume flow and thus a lower Reynolds number.
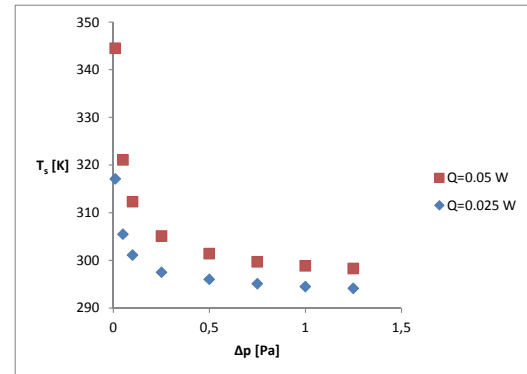

Figure 5. Solid plate temperature of optimized heat sink plotted against prescribed system pressure drop.
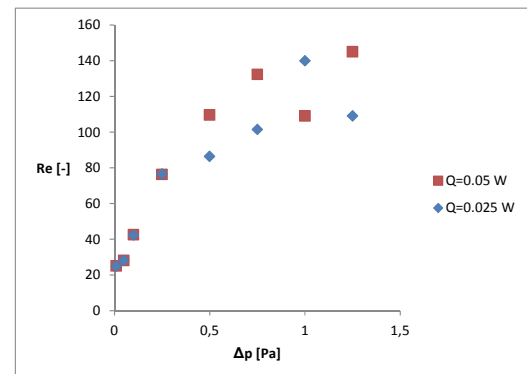

Figure 6. Reynolds number of optimized structures plotted against prescribed system pressure drop.

6. Conclusions

A 2D heat sink model with constant heat production and a three-field density-based topology optimization were implemented in COMSOL Multiphysics. The topology optimization was used to minimize the heat sink's temperature for a prescribed pressure drop and given heat production. Therefore, this work demonstrates that topology optimization of complex multiphysics systems can be implemented with comparatively little effort in COMSOL.

In a future work, transient and 3D validation simulations without symmetry condition will be

conducted with the optimized geometries to judge the physical validity of the obtained 2D optimization results.

7. References

1. J.D. Deaton, R.V. Grandhi, A survey of structural and multidisciplinary continuum topology optimization: post 2000, *Structural and Multidisciplinary Optimization*, **49**, 1-38 (2014)
2. E.M. Dede, T. Nomura, J. Lee, *Multiphysics Simulation*, Springer (2014)
3. M.P. Bendsøe, O. Sigmund, *Topology Optimization - Theory, Methods and Applications*, Springer Verlag, Berlin Heidelberg, 2003
4. F. Wang, B.S. Lazarov, O. Sigmund, On projection methods, convergence and robust formulations in topology optimization, *Structural and Multidisciplinary Optimization*, **43**, 767-784 (2011)
5. M. Jansen, B.S. Lazarov, M. Schevenels, O. Sigmund, On the similarities between micro/nano lithography and topology optimization projection methods, *Structural and Multidisciplinary Optimization*, **48**, 717 – 730 (2013)
6. M. Zhou, B.S. Lazarov, O. Sigmund, Topology optimization for optical projection lithography with manufacturing uncertainties, *Applied optics*, **53**, 2720-2729 (2014)
7. E.M. Dede, Multiphysics topology optimization of heat transfer and fluid flow systems, *Proceedings of the COMSOL Users Conference 2009 Boston, (2009)*
8. G.H. Yoon, Topological design of heat dissipating structure with forced convective heat transfer, *Journal of Mechanical Science and Technology*, **24** , 1225-1233 (2010)
9. T. Matsumori, T. Kondoh, A. Kawamoto, T. Nomura, Topology optimization for fluid-thermal interaction problems under constant input power, *Structural and Multidisciplinary Optimization*, **47**, 571-581 (2013)
10. G. Marck, M. Nemer, J.L.Harion, Topology optimization of heat and mass transfer problems: laminar flow, *Numerical Heat Transfer, Part B: Fundamentals,* **63**, 508-539 (2013)
11. J. Alexandersen, N. Aage, C.S. Andreasen, O. Sigmund, Topology optimisation for natural convection problems, International Journal for Numerical Methods in Fluids, **76**, 699-721 (2014)
12. T. Borrvall, J. Petersson, Topology optimization of fluids in Stokes flow, *International journal for numerical methods in fluids*, **41**, 77-107 (2003)
13. C. McConnell, G. Pingen, Multi-layer, pseudo 3D thermal topology optimization of heat sinks, *Proceedings of the ASME 2012 International Mechanical Engineering Congress, Houston, (2012)*
14. M. Stolpe, K. Svanberg, An alternative interpolation scheme for minimum compliance topology optimization, *Structural and Multidisciplinary Optimization*, **22**, 116-124, 2001
15. B.S. Lazarov, O. Sigmund, Filters in topology optimization based on Helmholtz-type differential equations, *International Journal for Numerical Methods in Engineering*, **86**, 765-781 (2011)
16. K. Svanberg, A class of globally convergent optimization methods based on conservative convex separable approximations, *SIAM journal on optimization*, **12**, 555-573 (2002)
17. O. Sigmund, J. Petersson, Numerical instabilities in topology optimization: a survey on procedures dealing with checkerboards, mesh-dependencies and local minima, *Structural optimization*, **16**, 68-75, (1998)

8. Acknowledgments