International Journal of Heat and Mass Transfer 116 (2018) 1108–1123

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

A "poor man's approach" to topology optimization of cooling channels based on a Darcy flow model

Xi Zhao [a], Mingdong Zhou [b,*], Ole Sigmund [c], Casper Schousboe Andreasen [c]

[a] School of Automobile Engineering, Dalian University of Technology, No. 2 Ligong Road, Dalian, China

[b] State Key Laboratory of Mechanical System and Vibration, Shanghai Key Laboratory of Digital Manufacture for Thin-walled Structures, Shanghai Jiao Tong University, No. 800 Dong Chuan Rd. Shanghai 200240, China

[c] Department of Mechanical Engineering, Technical University of Denmark, Building 404, Nils Koppels Allé, 2800 Kgs. Lyngby, Denmark

ARTICLE INFO

Article history:
Received 11 March 2017
Received in revised form 28 August 2017
Accepted 24 September 2017
Available online 17 October 2017

Keywords:
Darcy flow
Convective heat transfer
Cooling channels
Topology optimization
Length-scale control

ABSTRACT

A topology optimization methodology for optimizing cooling channels using an approximate but low-cost flow and heat transfer model is presented. The fluid flow is modeled using the Darcy model, which is a linear problem that can be solved very efficiently compared to the Navier–Stokes equations. The obtained fluid velocity is subsequently used in a stabilized convection–diffusion heat transfer model to calculate the temperature distribution. The governing equations are cast in a monolithic form such that both the solid and fluid can be modeled using a single equation set. The material properties: permeability, conductivity, density and specific heat capacity are interpolated using the Solid Isotropic Material with Penalization (SIMP) scheme. Manufacturable cooling-channel designs with clear topologies are obtained with the help of a pressure drop constraint and a geometric length-scale constraint. Several numerical examples demonstrate the applicability of this approach. Verification studies with a full turbulence model show that, although the equivalent model has limitations in yielding a perfect realistic velocity field, it generally provides well-performing cooling channel designs.

1. Introduction

Many engineering products and processes such as engines, batteries and CPUs are subject to heat loading from combustion processes or Joule heating. Other heat loads could be due to friction as in stamping or deep-drawing or come from hot solidifying material used in injection molding or direct heat conduction from the stamp in hot stamping processes. Most of this heat is harmful and it is important to transport this heat away from the working zone such that the final product complies with the tolerance requirements and to ensure a long lifetime of both product and tool. Besides this, the successful repeated and fast manufacturing of components also relies on the ability to control the local temperature of the process.

A standard method to remove heat is by letting a cooling liquid pass through channels in the heat generating component. A liquid cycling system would then transport the heat from the component to a heat exchanger. An effective cooling strategy depends on the distribution, periodicity and strength of potentially multiple heat sources. Amongst different processes, some may prefer a low temperature of the tool while others may benefit from a constant but higher temperature. Layout and shape of the cooling channels hence become crucial for the success of the manufacturing activities.

Much effort has already been invested by engineers and researchers in designing efficient cooling systems [9,16,22]. A typical optimization approach for cooling system design is by parametric optimization of channels in a predefined layout. Tan et al. [25] introduce a gradient-based cooling channel design methodology with parametrized geometric control points. Jarrett [17] uses the distance and thickness of channels as parameters to optimize heat transport in battery. Qiao [22] presents a 2D pipe section optimization design by using the boundary element method. Wang et al. [27] use a numerical experience based surrogate model for optimizing a 2D section design. Hu et al. [16] optimize a pipe design in a 3D model by taking the structural scale as parameters. Choi et al. [10] present a 3D channel design using a graphics pretreatment method to simplify the physical domain. However, such approaches do not allow introducing new channels or altering connectivity of channels during the optimization process.

* Corresponding author.
E-mail addresses: zhaoxivipdlut@mail.dlut.edu.cn (X. Zhao), mdzhou@sjtu.edu.cn (M. Zhou), sigmund@mek.dtu.dk (O. Sigmund), csan@mek.dtu.dk (C.S. Andreasen).

https://doi.org/10.1016/j.ijheatmasstransfer.2017.09.090
0017-9310/

The limitations of parametric optimization of cooling channels can be overcome by topology optimization [5], which is a systematic design approach for generating optimal material (here fluid) distributions. Originating from solid mechanics [4], this methodology has been extended to a wealth of other physical problems such as acoustics, electro-magnetics, heat transfer, fluid flow etc.

The general optimization of cooling channel systems involves flow simulation and convective heat transfer. Topology optimization was first applied to Stokes flow [6] and later extended to Navier–Stokes flow for low-to-moderate Reynolds numbers[12] by using a lubrication model approximation of flow between flat plates. An alternative interpolation model utilizing Darcy-Stokes flow was proposed by Guest [15] based on Brinkman's model [7] and Darcy's Law. In the lubrication approximation, a coefficient of viscous force is interpolated to control the flow. In the Darcy-Stokes model, the interpolation between solid and fluid is based on a porous flow assumption. The two models are related and they end up with similar structure where fluid flow in solid regions is penalized by an artificial force term proportional to the local velocity. The method was further extended to transport problems [3] in the form of passive mixer design. In recent years, problems including both forced and natural convection have attracted much attention in the quest for optimizing cooling profiles and heat exchangers. Dede [11] used a Brinkman model to optimize the cooling profile of jet impingement surfaces. Heat-sink devices were designed using a Brinkman penalization applied to the Stokes model in[18]. Yoon [28] focused on the forced convection heat transfer problem using Navier–Stokes equations to describe the fluid field. Alexandersen et al. [1,2] focused on cooling by natural convection in 2D and 3D problems. Zhou et al. [29] utilized a simpler engineering model for topology optimization of conductive and convective heat transfer problems. The latter uses the convective heat transfer coefficient to describe the convective thermal load and thus the overall system is solved very efficiently.

For practical cooling channel design, the most widely used simulation models are turbulent flow models coupled with convective heat transfer via the fluid velocity. Such complex models require non-linear solution schemes and boundary capturing meshing for simulation, which is time-consuming and thus unattractive for early stage conceptual design studies using topology optimization. Hence, there is demand for an efficient model to predict the fluid and temperature distributions using a computationally cheap but still sufficiently accurate method.

In this paper, a Darcy (potential) flow-based topology optimization approach is introduced for designing cooling channels. The temperature and heat transport is modeled using a stabilized convection–diffusion heat transfer model on top of the Darcy model. The modeling error between the Darcy and turbulent flow model is compared with the commercial software COMSOL. By parameter studies suitable relations between the material properties involved in the optimization process are found in order to minimize this error. The manufacturability of the resulting designs, i.e. clear black and white topology layouts with strongly enforced minimum length scale, are ensured using a combined projection and geometric constraint approach (c.f. [23,30]). The optimization problem is solved by the method of moving asymptotes (MMA)[24]. The optimized designs are verified with a complex thermo-fluidic model and show a similar trend in optimized performance but the accuracy of the temperature field cannot be guaranteed.

The remainder of the text is organized as follows. In Section 2, the Darcy-flow based convective heat transfer model is introduced and compared with a turbulent flow based model. The material properties of the solid and fluid phases are also studied and equivalent parameters for the simplified model are obtained. In Section 3.1, the material interpolation, filtering and projection methods are introduced. Topology optimization formulation and related constraints are discussed in Section 3.2. Section 4 presents several numerical examples with associated discussion. Conclusions are given in Section 5.

2. Physical model

Fig. 1 shows a generic cooling system design problem. The physical design domain Ω consists of a solid phase domain Ω_s, a water phase domain Ω_w as $\Omega = \Omega_s \cup \Omega_w$. The boundary of the domain Ω is divided into two parts, an outlet boundary Γ_1 and an inlet boundary Γ_2. Boundary conditions such as prescribed pressure, $p = p_0$ or prescribed velocity, $\mathbf{u} \cdot \mathbf{n} = u_0$, can be applied on both inlet and outlet, where $\mathbf{n}$ is the normal vector of boundary Γ_1 or Γ_2. The temperature at the inlet boundary is set constant $T = T_0$. The remaining part of the boundary $\partial\Omega \setminus (\Gamma_1 \cup \Gamma_2)$ is ano-flow ($\mathbf{u} \cdot \mathbf{n} = \mathbf{0}$) and isolating ($\frac{\partial T}{\partial \mathbf{n}} = 0$) boundary. The heat source in Ω_h represents the area that generates the heat and gives rise to the cooling problem. This domain may be included for design or kept as a passive solid zone depending on the problem approached.

2.1. Heat transfer model with Darcy flow

An internal flow problem is generally modeled using the incompressible steady-state Navier–Stokes equations:

$$\rho\mathbf{u} \cdot \nabla\mathbf{u} = -\nabla p + \mu\nabla^2\mathbf{u} - \rho\mathbf{b}, \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0, \tag{2}$$

where $\mathbf{u}$ denotes the velocity field and p the pressure, μ is the dynamic viscosity, ρ is the mass density and the vector $\mathbf{b}$ is the body force per unit mass. Viscosity of the cooling fluids depends on temperature in general. The equations may be solved assuming isothermal conditions as the temperature dependence is rather weak for small temperature variations.

For active cooling strategies based on pumping of the fluid through the channels, a turbulent flow model is generally needed to model the internal fluid mixing as well as the thin boundary layer where most heat is exchanged. The velocity is determined by friction in turn resulting in a pressure loss (gradient) across the component. In practice, turbulence models such as Reynolds-averaged Navier–Stokes (RANS) can be utilized to simulate this flow. However, solving the non-linear Navier–Stokes equations is difficult in a topology optimization setting and very time-consuming. For early conceptual design of cooling layouts, fast simulation and low turnaround time are critical and hence a computationally efficient model with acceptable accuracy is required.

Considering that the velocity profile of a fully turbulent flow is much flatter than the parabolic fully developed laminar profile, we propose to approximate the flow as an inviscid one by a Darcy potential flow model:

$$\mathbf{u} = -\frac{\kappa}{\mu}\nabla p, \tag{3}$$

where the velocity is proportional to the pressure gradient by the ratio of the permeability κ and the viscosity μ. Inserting this into the incompressibility condition and by ignoring the body force term yields the following model:

$$\nabla \cdot \left(-\frac{\kappa}{\mu}\nabla p\right) = 0. \tag{4}$$

The velocity profile of a Darcy flow in a confined channel is uniform and flat without any boundary effects at all. However, by careful selection of the artificial permeability of fluid and solid