Multiphysics Topology Optimization of Heat Transfer and Fluid Flow Systems

Ercan M. Dede*[1]
[1]Technical Research Department, Toyota Research Institute of North America
*Corresponding author: Toyota Motor Engineering & Manufacturing of North America, Inc., 1555 Woodridge Ave., Ann Arbor, MI, 48105. Email address: eric.dede@tema.toyota.com

Abstract:

This paper is focused on topology optimization of heat transfer and fluid flow systems for multiphysics objectives. Specifically, COMSOL Multiphysics software is coupled with a method of moving asymptotes optimizer in a custom COMSOL / MATLAB script. Various physical process including conduction, convection-diffusion, and Navier-Stokes flow are considered. To illustrate the method, a standard pure heat conduction problem is first presented in two dimensions followed by an extension of the problem to three dimensions. More complex physics are then examined in the optimization process for a three-terminal heat transfer and fluid flow device. General applications and limitations of the methodology are discussed.

Keywords:

Multiphysics, topology optimization, thermal, fluid

1. Introduction

Advanced electrical machine design requires simulation and optimization of systems for multiple physical processes. Within this active engineering field several types of multiphysics problems may be defined including those related to heat transfer and fluid flow. While determining the ‘optimal’ structure for a standalone physical process may be straightforward, defining structures for objectives involving many physics is more challenging. Accordingly, the engineering community is continuously examining the feasibility of various optimization techniques for such tasks.

One potential technique is structural topology optimization, which is a broad research area with a variety of interesting applications [1]. Basically, topology optimization consists of an iterative loop in which finite element analysis, sensitivity analysis, and optimization steps (to update design variables) are performed [2]. In this study COMSOL software is used in conjunction with a method of moving asymptotes (MMA) optimizer [3] to streamline this process. A similar set of computational tools has been successfully applied by previous researchers to the optimization of various single physics problems [4,5]. A primary advantage of the approach is that the designer may exploit COMSOL for the finite element and sensitivity analysis portions of the problem in a custom scripting environment. Additionally, the designer may make efficient use of their time both for model generation and post processing of results.

This paper is focused on the use of COMSOL Multiphysics with a MMA optimizer in a custom MATLAB script for topology optimization of heat transfer and fluid flow problems. The optimization process is briefly reviewed in Section 2. A description of a two-dimensional (2-D) single physics pure heat conduction benchmark problem is presented in Section 3 along with a similar problem in three-dimensions (3-D). These problems were examined to validate the optimization process in the COMSOL environment, and optimal topologies are provided in Section 4. A 2-D thermal / fluid multiphysics optimization problem is then described in Section 5 followed by results in Section 6. Discussion and conclusions are given, respectively, in Sections 7 and 8.

2. Topology Optimization – Brief Review

The material distribution method is a common approach to topology optimization. A concise explanation of the process is given here. The reader is referred to the literature for an in-depth discussion of the topic [1,2].

To optimize the topology of a structure it is first typically discretized into many finite elements as part of a computational analysis. The material distribution method is then used to find the optimal topology of the structure for a given objective and constraints by assigning each element an individual density value. These density values may be associated with a material physical parameter such as the isotropic thermal conductivity and are interpolated from 0 (no material) to 1 (solid material) using various 'penalization' schemes that influence material distribution. The final element density values are found through an iterative loop that involves repeated evaluation of an objective function and gradients [1].

The implementation of the above process in COMSOL Multiphysics varies slightly as described in [4,5]. Nonetheless, the general process is still applicable to the solution of a standard nonlinear optimization problem [3],

$$\begin{aligned} &\text{minimize} && f_0(\mathbf{x}) && \\ &\text{subject to} && f_i(\mathbf{x}) \le 0 && i = 1, \dots, m \ , \\ & && x_j^{\min} \le x_j \le x_j^{\max}, && j = 1, \dots, n \end{aligned} \qquad (1)$$

where f_0 is the objective function, f_i are behavior constraints, m is the number of constraints, and $\mathbf{x}$ is a vector of n design variables, x_j.

For a single physics problem (e.g. pure conduction), the objective function may be as straightforward as minimizing the mean temperature of the design domain subject to a given volume constraint, where the material thermal conductivity is interpolated from zero to a predetermined maximum value. For a multiphysics problem, the objective function may consist of separate parts related to different physical processes (e.g. heat transfer plus fluid flow).

3. Single Physics – Heat Conduction

A single physics heat conduction optimization study was performed for validation of the COMSOL / MMA optimization process. The governing equations and the optimization objective function for this study are provided in Section 3.1. Descriptions of the model geometries, boundary conditions, and loads are given in Section 3.2. The COMSOL Heat Transfer Module was used within the MATLAB environment to generate and evaluate all models.

3.1 Governing Equations

For steady state pure conduction Fourier's law governs heat transfer,

$$-\nabla \cdot (k \nabla T) = Q \ , \qquad (2)$$

where k is the thermal conductivity of the solid material, T is the temperature state variable, and Q is the volumetric heat generation.

The general objective function, A_1, chosen for this single physics benchmark problem is equivalent to minimizing the mean temperature of the design domain, Ω, subject to constant heat generation,

$$A_1 = \int_\Omega k (\nabla T)^2 \, \mathrm{d}\Omega \ . \qquad (3)$$

The objective function, Eq. (3), may be expressed in three dimensions as a global expression within a custom script: '`fem.globalexpr = {'A','(.001+0.999*rho^penal)*k*(Tx*Tx+Ty*Ty+Tz*Tz)'}`'. In this expression '`rho`' is the density design variable, '`penal`' is the penalization power for interpolation, '`k`' is the material isotropic thermal conductivity which varies between zero and unity, and '`Tx`,' '`Ty`,' and '`Tz`' are the partial derivatives $\partial T / \partial x$, $\partial T / \partial y$, and $\partial T / \partial z$, respectively.

3.2 Computational Model Descriptions

Determining the optimal material distribution for a square 2-D design domain subject to pure conduction, Figure 1, is a standard optimization problem available in the literature [1].

The temperature at the center of the left edge of the domain in Figure 1 was set to zero to represent a heat sink. The remaining edges of the domain were considered adiabatic (i.e. zero heat transfer). Constant uniform heat generation was assumed throughout the domain which was meshed using approximately 10,000 quadrilateral elements.

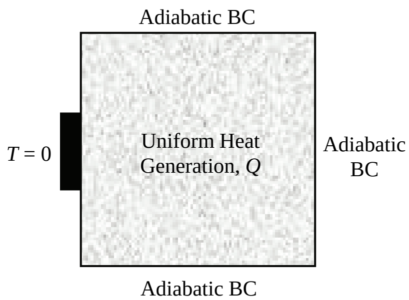

Figure 1: 2-D topology optimization design domain and boundary conditions for pure heat conduction.

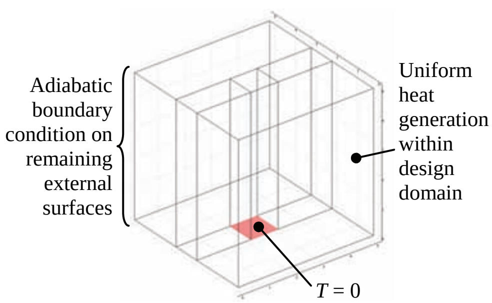

Figure 2: 3-D topology optimization design domain and boundary conditions for pure heat conduction.

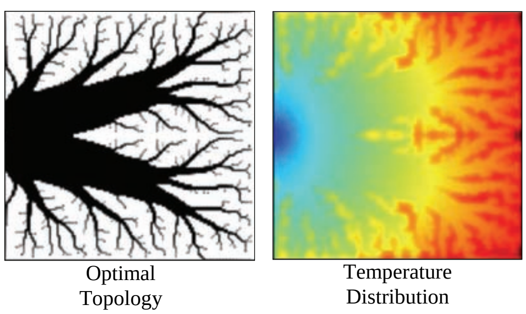

Figure 3: Optimal topology (shown by dark regions) and temperature distribution for 2-D design domain.

The extension of this problem to 3-D is relatively straightforward with the help of the COMSOL Multiphysics graphical user interface. In Figure 2 the 3-D design domain is shown partitioned into separate regions to facilitate boundary condition assignment. The highlighted lower boundary in Figure 2 represents the heat sink region. All other boundaries are again adiabatic, and the domain is subject to a uniform heat source. The 3-D design domain was meshed using approximately 8,000 hexahedral elements.

A maximum solid volume fraction of 0.4 was used in the optimization process for both 2-D and 3-D models.

4. Single Physics – Optimization Results

The optimal topology and temperature field for the 2-D model are shown in Figure 3. These results are consistent with those found in the literature [1]. For the 3-D model, a self-similar 'branching,' or dendritic structure is also obtained; see Figure 4. The 3-D optimal topology was reconstructed in MATLAB using the Image Processing Toolbox while temperature slices through the design domain were obtained using standard COMSOL post-processing. Interestingly, the evolution of the vascular structures in Figures 3 and 4 appears to follow constructal law, which allows heat to flow via the easiest path from source to sink [6].

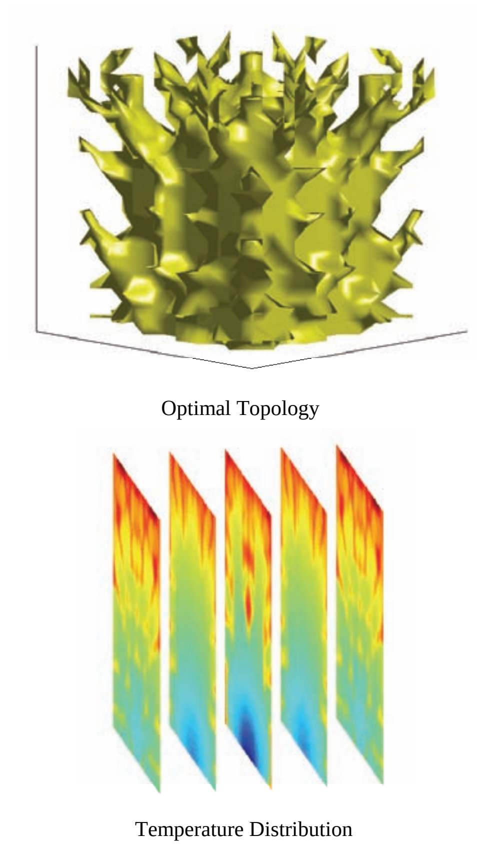

Figure 4: Optimal topology and temperature distribution slices of 3-D design domain.

5. Multiphysics – Fluid Flow plus Heat Transfer

In Section 5.1 the governing equations and objective function are provided for a general thermal / fluid problem. A three-terminal 2-D model is defined in Section 5.2. The General PDE mode was used to generate and evaluate the model.

5.1 Governing Equations

Following [4,7], governing equations for steady state flow in an idealized porous medium are,

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

$$\rho(\mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla P + \nabla \cdot \left\{\eta\left[\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}\right]\right\} - \alpha \mathbf{u}\,. \tag{5}$$

Eq. (4) represents the fluid incompressibility constraint, and Eq. (5) describes Navier-Stokes fluid flow. In these expressions ρ and η are the fluid density and dynamic viscosity, respectively. The inverse permeability of the porous medium, α, is assumed to be approximately valid for an actual porous medium, per [4]. The state variables include the fluid pressure, P, and velocity field terms in the vector, **u**.

Additionally, the governing equation for steady state convection-diffusion heat transfer is,

$$\rho C(\mathbf{u} \cdot \nabla T) = \nabla \cdot (k \nabla T) + Q\,, \tag{6}$$

where C represents the heat capacity and k is the thermal conductivity of the fluid.

A dual objective function, A_2, was implemented to optimize for both heat transfer and fluid flow. Specifically, the objective was specified to minimize the mean temperature and total fluid power dissipated in the system,

$$A_2 = w_1 B + w_2 C \quad \text{, where} \tag{7}$$

$$B = \int_{\Omega} \left\{k(\gamma)(\nabla T)^2 + \rho C\left[\mathrm{T}(\mathbf{u} \cdot \nabla T)\right]\right\} \mathrm{d}\Omega\ ; \tag{8}$$

$$C = \int_{\Omega} \left[\frac{1}{2}\eta \sum_{i,j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)^2 + \sum_i \alpha(\gamma) u_i^2\right] \mathrm{d}\Omega\,. \tag{9}$$

Eq. (7) is implemented as a global expression as previously described in Section 3.1. In this expression B is proportional to the mean temperature of the design domain for constant heat generation, Q, while C is related to the total flow power dissipated in the fluidic system [4].

The terms w_1 and w_2 in Eq. (7) are weighting values that scale the respective thermal and fluid portions of the objective function. ‘Tuning’ these values was found to assist in convergence and modifies the resulting optimal topology by affecting the dominance of one physical process relative to another. For simplicity, these weighting values were selected manually.

To determine the optimal steady state fluid flow and channel layout the thermal conductivity and inverse permeability of the porous medium were, respectively, interpolated using the penalty method from [1] and the convex interpolation scheme from [4]. These effective properties (i.e. k and α) were interpolated via a main design parameter, γ, which varied from 0 (minimally-porous, non-conductive solid) to 1 (conductive fluid).

5.2 Computational Model Description

To illustrate the multiphysics optimization process a 2-D three-terminal structure, separated into four subdomains, was considered; see Figure 5. A domain roughly square in size and having a height a little larger than its width was selected to provide slightly greater distance between the two fluid outlet terminals. The design domain was meshed with approximately 6,200 quadrilateral elements.

Fixed temperature, parabolic normal fluid flow was assumed at the single device inlet. Convective flux, zero pressure normal flow was assumed at both outlets. No-slip adiabatic boundary conditions were enforced on all external walls of the device. Uniform heat generation, Q, and the main design parameter, γ, were specified only on the primary design domain.

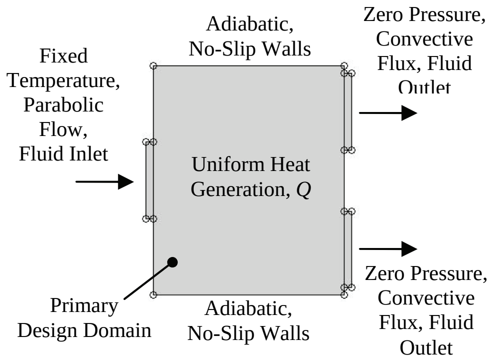

Figure 5: 2-D topology optimization design domain and boundary conditions for fluid flow and heat transfer.

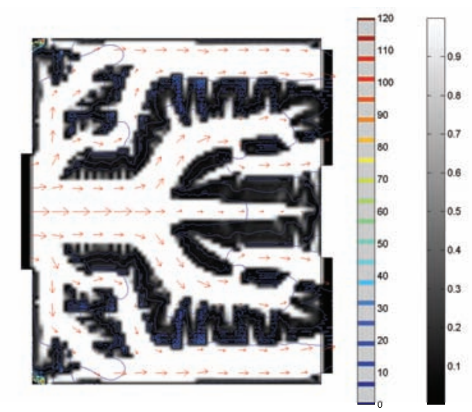

Figure 6: Optimal topology, temperature contours, and fluid flow vectors for three-terminal device (color contour bar refers to temperature contours; grayscale bar refers to solid – 0 / fluid – 1 material distribution).

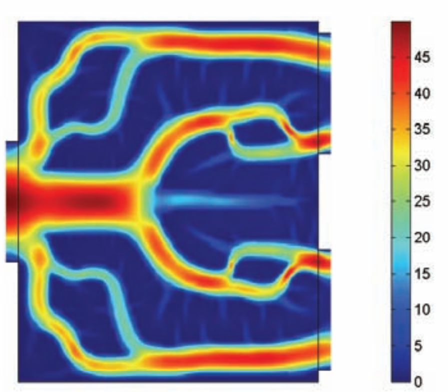

Figure 7: Fluid flow velocity contours for optimal heat transfer and fluid flow.

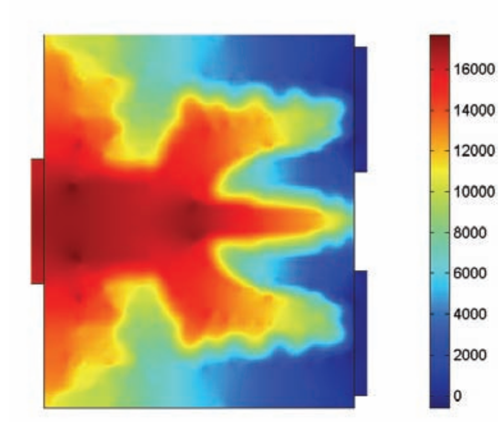

Figure 8: Pressure contours for optimal heat transfer and fluid flow.

The inlet fluid Reynolds number was set arbitrarily to Re ≈ 50. A minimum Darcy number of 10^{-4} was selected to represent a relatively impermeable solid [4]. Fluid density, heat capacity, and dynamic viscosity were all set to unity. The solid volume fraction constraint was set 0.6 for designs exhibiting a lesser amount of solid material.

6. Multiphysics – Optimization Results

Figure 6 shows the channel topology, temperature contours, and fluid flow vectors obtained for the three-terminal device optimized for minimum average temperature and power dissipation. Observe that the solid (darker) regions have increased temperature relative to the free fluid flow (lighter) areas with hot spots occurring in the domain corners. This thermal behavior is expected for minimally porous, non-conductive solid material versus conductive fluid, respectively.

The optimal fluid flow velocity and pressure contours are also shown for reference in Figures 7 and 8, respectively. In Figure 7 regions of less porous semi-solid material are shown in dark blue versus fluid flow channels in lighter colors; refer to the contour bar shown in the figure. This optimal fluid flow topology is an artifact of the multiple objectives in Eq. (7). Specifically, there is a tradeoff between a fluid distribution that minimizes the average temperature of the domain, Figure 7, and flow resistance (i.e. pressure drop), Figure 8.

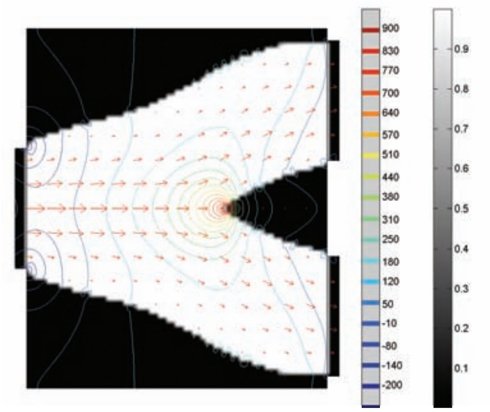

Figure 9: Optimal 'Y-branch' topology with pressure contours and fluid flow vectors for minimum flow power dissipation only (i.e. obtained with w_1 = 0 and w_1 = 1 in Eq. (7)).

Note that setting w_1 = 0 and w_2 = 1 in Eq. (7) leads to a simple 'Y-branch' topology and a reduced pressure drop, Figure 9, for minimum fluid power dissipation. Thus, the relatively large pressure drop observed in Figure 8 is an artifact of a larger weighting given to the thermal portion of the objective function, *B*, in Eq. (7). Thus, the topology in Figure 6, obtained for dual heat transfer and fluid flow objectives, is a logical superposition of the dendritic structural characteristics seen in Figure 3 and the straightforward 'Y-branch' structure in Figure 9. The conclusion is that the relative values of w_1 and w_2 in Eq. (7) have an important effect in 'tuning' the final result obtained through the optimization process.

7. Discussion

The multiphysics optimization process described in this paper may be applied to a variety of heat transfer and fluid flow problems. Additional physics including, for example, both static and dynamic structural loading may be incorporated into the general process. Moreover, the advantage of using an MMA optimizer is that it has been shown to be effective in handling optimization problems with multiple constraints.

The primary limitation in applying this method to a broader range of three dimensional problems continues to be the computational time required for each iterative optimization step, which involves both a multiphysics finite element simulation plus sensitivity analysis. Additionally, the synthesis of the final optimization result into an actual physical structure is typically challenging. Nonetheless, as computing resources continue to improve, the use of similar methods for 3-D structural design will likely have a significant impact on the research and development process for advanced electrical machines and assist in reducing design cycle time.

8. Conclusions

The application of gradient based topology optimization within COMSOL Multiphysics via a MMA optimizer was presented. An initial single physics pure heat conduction problem was selected to evaluate a custom MATLAB script. Self-similar branching structures were obtained for this benchmark problem both in 2-D and 3-D. The method was then extended to multiple physical processes including convection-diffusion and Navier-Stokes flow with corresponding objectives. The approach was then applied to optimize a 2-D three-terminal device having characteristics attributable to the various physics involved.

Despite additional computational time needed for larger 3-D structures, this approach provides interesting starting points for synthesizing effective thermal / fluid structures. Future research should focus on automating the weighting strategy for multiple objectives to better interrogate a prospective design space. The application of this computational method to the design of various vehicle systems is also a logical focus for future work.

9. References

1. M.P. Bendsoe and O. Sigmund, *Topology optimization*, 2nd Edn., pp. 71-272, Springer-Verlag, Berlin (2004).
2. O. Sigmund, A 99 line topology optimization code written in MATLAB, *Structural and Multidisciplinary Optimization*, **21**, pp. 120-127 (2001).
3. K. Svanberg, The method of moving asymptotes – a new method for structural optimization, *International Journal for*

Numerical Methods in Engineering, **24**, pp. 359-373 (1987).

4. L.H. Olesen, F. Okkels, and H. Bruus, A high-level programming-language implementation of topology optimization applied to steady-state Navier-Stokes flow, *International Journal for Numerical Method in Engineering*, **65**, pp. 975-1001 (2006).

5. B. Lemke, Z. Liu, and G. Korvink, Implementation of structural topology optimization in COMSOL, *COMSOL Users Conference*, Frankfurt, Germany (2006).

6. A. Bejan and S. Lorente, Natural design with constructal theory, *Mechanical Engineering*, **13**, pp. 44-48 (2009).

7. J.N. Reddy and D.K. Gartling, *The finite element method in heat transfer and fluid dynamics*, 2nd Edn., pp. 7-10, CRC Press, Boca Raton (2000).