Level-set based Topology Optimisation of Convectively Cooled Heatsinks

M. Santhanakrishnan*[1], T. Tilford[1], C. Bailey[1]
1.Department of Mathematical Sciences, The University of Greenwich, London, UK
*m.santhanakrishnan@greenwich.ac.uk

Abstract:
The level set based topology optimisation technique is applied to optimise a 2D and 3D convectively cooled heatsink. The approach is used to determine the design providing minimal thermal compliance and minimal viscous dissipation. The optimisation has been performed utilising the COMSOL multiphysics for solution of physics and sensitivity analysis and the MATLAB Livelink functionality for level set advection and reinitialisation. This paper describes the details of implementation, the topology optimisation model and presents results obtained using this formulation.

Keywords: Level set, Topology optimisation, Heat sink, Forced convection, Re-initialisation

1.0 Introduction

A heat sink is a passive cooling device which transfers the heat received from the heat source to adjacent fluid medium through forced or natural convection. Many electronic devices, such as CPU, GPU, power transistors and LEDs generate significant levels of heat energy which must be efficiently dissipated in order to ensure reliable operation.

Topological optimisation (TO) is a mathematical approach that optimises material layout within a given design space, for a given set of constraints such that the resulting layout meets a prescribed set of performance objectives [1]. The technique has primarily been used for structural optimisation problems but it is been applied to various physical problems. The two most prevalent TO approaches are density based method and Level-set methods (LSM) [2], with the latter preferred due to the ability to sharply capture inter-material interfaces.

COMSOL Multiphysics software has been successfully used to apply LSM TO for design optimisation in a variety of physical problems. In this study we use COMSOL Multiphysics solver combined with MATLAB Livelink for performing the LSM TO with re-initialisation of the level set functions at regular iteration intervals to enhance accuracy.

In this study Two & Three dimensional heat sinks will be developed for two different solid to fluid thermal conductivity ratios for the objectives of minimum thermal compliance (TC) and viscous dissipation (VD). The rest of the paper details the LSM TO formulation in Section 2, Numerical Implementation in Section 3 and Computational details in Section 4. Results of the study and discussion are given in Section 5 and Conclusions are given in Section 6.

2.0 Level set based TO formulation

The seminal papers on LSM TO were written by Allaire [3] and Wang [4], where they applied this technique for the optimisation of structural members. Challis and Guest [5] applied this technique for the optimisation of Stokes flow problems. Subsequently this technique has been extended by many researchers to various fields.

In this technique, Signed Distance Function (SDF) is used as level set function (LSF). Positive SDF (ψ) is considered to represent solid and negative SDF is considered to represent fluid (Figure 1). This is enforced by the ersatz projection approach [3], using Heaviside function.

$$\psi = \begin{cases} = 0\ \forall x \in \partial\Omega\ (\text{boundary}) \\ > 0\ \forall x \in \Omega^+\ (Solid\ region) \\ < 0\ \forall x \in \Omega^-\ (Fluid\ region) \end{cases} \quad (1)$$

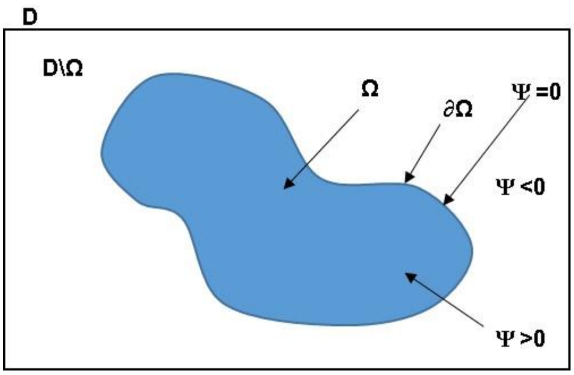

Figure 1 Design domain and level set function

Brinkman's porosity term (α) is used to differentiate solid and liquid and it is modelled as below.

$$\alpha = (\alpha_{max} - \alpha_{min})*H + \alpha_{min} \quad (2)$$

Where, H is Heaviside function, which take unit value when LSF is positive and takes zero value when LSF is negative and it has smooth transition between the two levels in order to enable differentiability. Derivative of Heaviside function is δ function whose expression is also given below.

$$H(\psi) = \frac{1}{2} + \frac{15}{16}\left(\frac{\psi}{h}\right) - \frac{5}{8}\left(\frac{\psi}{h}\right)^3 + \frac{3}{16}\left(\frac{\psi}{h}\right)^5 \quad (3)$$

$$\delta(\psi) = \frac{15}{16h}\left(1 - \left(\frac{\psi}{h}\right)^2\right)^2 \quad (4)$$

Where α_{max} =1e4 and α_{min} =0.01

Evolution of the LSF represents evolution of shape and this evolution with respect to time is governed by Hamilton-Jacobi (HJ) equation, given in (5). This equation is marched in time to convect the LSF in the decreasing direction of objective value. This is done by taking the velocity of convection equal to shape sensitivity.

HJ equation: $$\frac{\partial\psi}{\partial t} = V_n|\nabla\psi| \quad (5)$$

This equation is solved using an explicit first order upwind scheme. The time step for the marching should satisfy the CFL criterion for stability [3]. Every time the physical problem is solved, the HJ equation is marched in time several time steps in order to obtain new shape or new level set function. The MATLAB code (TOPLSM) written by Wang [6] demonstrates the various steps involved in level set based topology optimisation for simple structural mechanics problems.

The formulation of the approach requires the gradient of the LSF has to be unity. During LSM boundary convection this gradient may vary from unity and result in in-accuracy of boundary definition. In order to overcome this, the Eikonal equation (6) is solved to re-initialise the LSF [3]. The unsteady equation is time marched till steady state is obtained, the steady state ensures the gradient of level set equals 1.

$$\frac{\partial\psi}{\partial t} + w.\nabla\psi = S(\psi o)$$

$$w = S(\psi o)\frac{\nabla\psi}{|\nabla\psi|} \quad (6)$$

Where S is smoothed sign function $S(\psi)=\begin{cases}1\\-1\end{cases}$

$$S(\psi)_{i,j} = \frac{\psi_{i,j}}{\sqrt{{\psi_{i,j}}^2 + hmesh^2}} \quad (7)$$

Gradients calculated through forward and backward difference formulas are used to solve the equation. The difference formula used for time marching is given below.

$$a = D_x^-\psi_{i,j} = \frac{\psi_{i,j} - \psi_{i-1,j}}{h}$$

$$b = D_x^+\psi_{i,j} = \frac{\psi_{i+1,j} - \psi_{i,j}}{h}$$

$$c = D_y^-\psi_{i,j} = \frac{\psi_{i,j} - \psi_{i,j-1}}{h}$$

$$d = D_y^+\psi_{i,j} = \frac{\psi_{i,j+1} - \psi_{i,j}}{h}$$

$$G(\psi)_{i,j} = \begin{cases} \sqrt{\max((a^+)^2,(b^-)^2) + \max((c^+)^2,(d^-)^2)} - 1 & if\ \psi_{i,j}^0 > 0 \\ \sqrt{\max((a^-)^2,(b^+)^2) + \max((c^-)^2,(d^+)^2)} - 1 & if\ \psi_{i,j}^0 < 0 \\ 0 & otherwise \end{cases}$$

$$\psi_{i,j}^{N+1} = \psi_{i,j}^N - \Delta t\, S\left(\psi_{i,j}^0\right) G\left(\psi_{i,j}^N\right) \quad (8)$$

3.0 Numerical Implementation

The topology optimisation of heatsinks using Level set method with re-initialisation is implemented following the works of Liu [7], Kawamoto [8] and Deng [9]. The works of Liu and Kawamoto describe the coupled LSM TO formulation within COMSOL multiphysics, whereas Deng has used COMSOL Multiphysics for solving the physics and MATLAB Livelink for solving the HJ equation and for re-initialisation. The paper follows the approach of Deng [9] but applied to the challenge of heat sink design. Some of the notable works on heat sink design using topology optimisation are by Yoon [10], Alexanderson [11] using Density method and works of Yaji [12] and Coffin [13] using Level set method. Yaji designed liquid channel cooled heat sinks and Coffin designed convectively cooled heat sinks using level set with Extended Finite Element Method geometry mapping.

The various steps involved in the process are given in Figure-2, in which steps enclosed within the dashed line box are carried out in COMSOL Multiphysics and rest of the steps are carried out in MATLAB Livelink.

First, the design domain is initialised with some initial guess of LSF in MATLAB and it is exported to COMSOL Multiphysics. For the heat sink design, the points within the design domain are differentiated into solid or fluid by interpolating the properties viz, thermal conductivity, Cp, density and material impermeability. This in turn depends on the sign of level set function and Heaviside function as given in Table-1.

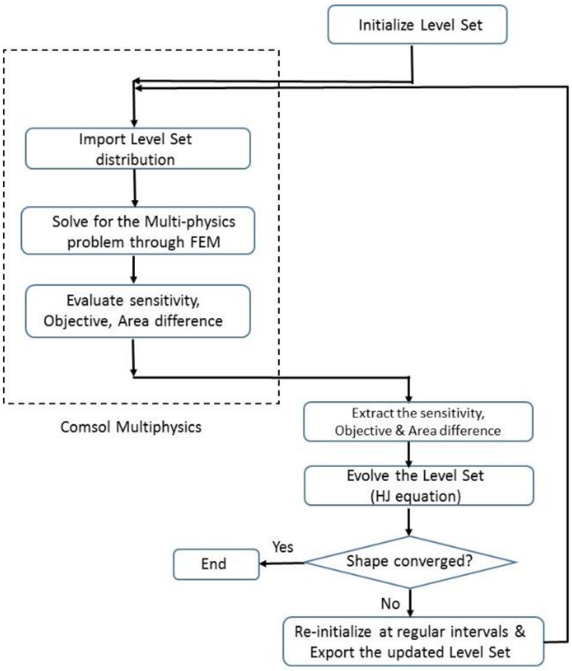

Figure 2 Level set topology optimisation procedure

Name	Expression
Kgam	$(K_s - K_f)*H + K_f$
Cpgam	$(Cp_s - Cp_f)*H + Cp_f$
ρgam	$(\rho_s - \rho_f)*H + \rho_f$

Table 1 Interpolation of thermal properties

The heat sink optimisation problem for two different objectives are stated below.

Objective TC: $min \int_{\Omega} \; kgam * (\nabla T)^2 d\Omega$

Objective VD: min $\quad \mu \int_{\Omega} (\frac{\partial u_i}{\partial x_j})^2 \, d\Omega$ (9)

Subjected to,

$$\rho Cp(u.\nabla T) = \nabla.(k\nabla T) + Q \qquad (10)$$

$$(\nabla.u) = 0 \qquad (11)$$

$$\rho(u.\nabla u) = -\nabla p + \nabla.\{\mu\{\nabla u + (\nabla u)^T\}\} - \alpha u \qquad (12)$$

H(Ψ)u=0 (13)

Volume constraint = 0.40*Design volume (14)

In order to import the LS in COMSOL from MATLAB, “Interpolation” Function is used. Since the mesh remains same and only the level set function is evolving, no accuracy will be lost due to interpolation. By solving the physical problem in COMSOL, the shape sensitivity is calculated and then it is retrieved in MATLAB Livelink using command ‘mpheval’, the syntax is given below.

mpheval(model, 'shapesens','dataset','dset2','selection',3)

where ‘shapesens’ is the variable name, ‘dset2’ is dataset number and ‘3’ is the design domain number. No-slip is imposed by initialising velocity equal to zero in solid regions i.e., where H equal to 1.

For TC objective, the velocity term in the HJ equation (5) is equal to,

$$Vn = \left[Kgam * \left((\frac{\partial T}{\partial x})^2 + (\frac{\partial T}{\partial y})^2 + (\frac{\partial T}{\partial z})^2\right) + \lambda + \Lambda(Volume\ difference)\right] \qquad (15)$$

Where the first term on the right hand side denotes the shape sensitivity for the thermal compliance objective, λ is the Lagrangian multiplier and Λ is the area penalty factor.

For VD objective, shape sensitivity is,

$$Vn = 0.5 * \mu * \left((\frac{\partial u_i}{\partial x_j}) + (\frac{\partial u_j}{\partial x_i})\right)^2 + \lambda + \Lambda(Volume\ difference) \qquad (16)$$

The Lagrangian multiplier and Area penalty factor are updated as follows.

$$\lambda_k = \lambda_{k-1} - \Lambda_{k-1}\,(Voume\ Difference) \qquad (17)$$

$$\Lambda_k = \frac{1}{\beta}\Lambda_{k-1} \qquad (18)$$

The value of β is 0.9, and the same value is used for all simulations irrespective of the objective of the study but value of Lagrangian multiplier λ, and the area penalty factor Λ, are chosen differently for different objectives. The reason for choosing different value is the difference in magnitude of the objective values. Suitable value of these factors are chosen by trial and error method.

The level sets are re-initialised at regular intervals (every 4th iteration or 5th iteration) to maintain their slope. It is noted that due to re-initialisation, the mean line of the boundary is slightly moved or there may be a phase lag. Level set distribution before and after re-initialisation at one instant is shown in Figure-3. Topology gradient term is not implemented in the algorithm so no new holes are nucleated in the design domain.

The evolved level sets are again feedback to the COMSOL Multiphysics and this procedure is repeated till convergence.

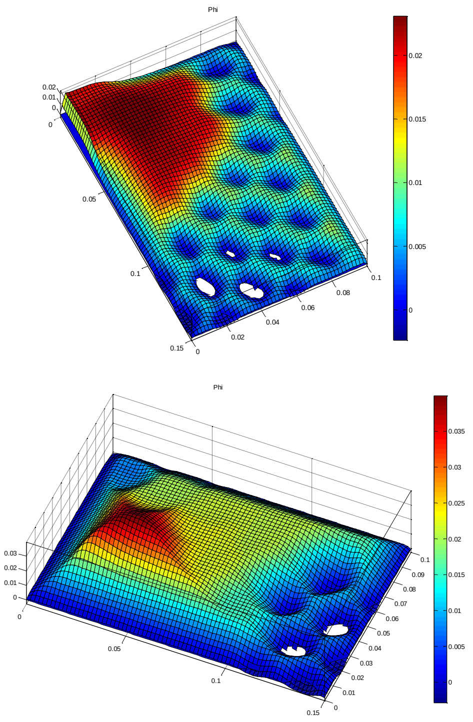

Figure 3 Level set function before and after re-initialisation

4.0 Computational details

Design domain is rectangular in shape, with heat source at the bottom of the domain and liquid convection injected from the top of computational domain as shown in Figure-4. The two sides of the computation domain act as outlet.

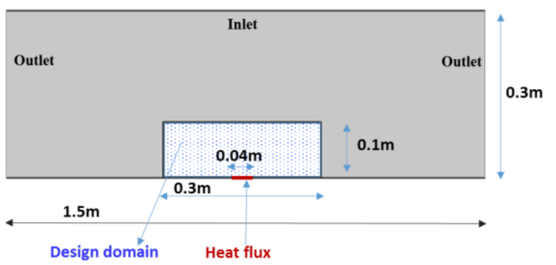

Figure 4 Computational domain details

The design domain is discretised with 150x50 rectangular elements. The initial level set used for the computation is series of circles. The level set function is evolved on a grid mesh with ghost elements, these elements surround the 4 sides of design domain. '*Heat transfer in fluids*' and '*Laminar flow*' modules of COMSOL are used for computation. A liquid flow of velocity 0.002m/s and temperature 293K is applied at the Inlet. The inlet velocity corresponds to a Reynolds number of 600 and a heat flux of 700W/m^2 is specified as heat source in the bottom wall and zero pressure boundary condition is applied at the outlet. TO is carried out for two different material sets i.e., fluid to solid conductivity ratios. The material properties are given in Table-2.

Parameter	Value
K_f / K_s	0.001 & 0.1
ρ_f / ρ_s	1000/8920
Cp_f / Cp_s	4184/385

Table 2 Material properties

Computational domain used for 3D study is shown in Figure-5. The computational domain considered is 1 quadrant of the total domain, making use of symmetry boundary condition on the two sides. The design domain is a cube of side 0.1m length and it is discretised by 43x43x43 mesh elements. Heat flux of 1000W/m^2 is applied at the bottom corner of area 1.353e-4 m^2 of the design domain base and a fluid flow of velocity 4e-5m/s is applied at the top surface of the computational domain. The volume fraction of solid material is constrained at 25%.

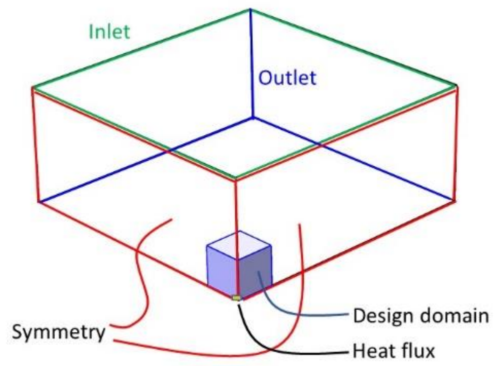

Figure 5 Three dimensional computational domain

Linear elements or discretisation is used for both velocity and pressure along with stream wise diffusion stabilization for finite element solution. In two dimensional case, governing equations are solved in coupled way and for three dimensional case equations are solved in segregated way. Simulation meeting the area constraint and the objective not varying significantly is considered as converged.

5.0 Results and Discussion

Considering only TC as objective, TO is carried out for 2 fluid to solid conductivity ratios k_f/k_s=0.001 and 0.1. Since the flow Reynolds numbers are low (Re600), it is expected that at least for later case, the convective heat transfer will play a comparable role to conduction. Results obtained are given in Section

5.1 and 5.2 and later section describe the result for VD minimisation case and combination of TC and VD objectives.

5.1 Heat sink of higher solid conductivity k_f/k_s=0.001 case:

The optimized shape for higher solid thermal conductivity case, resemble like a tree shape and it is shown in Figure-6. Temperature is uniformly distributed throughout the design domain expect near the peripheries. Convergence of Lagrange Multiplier, Area difference and Thermal compliance are shown in Figure-7.

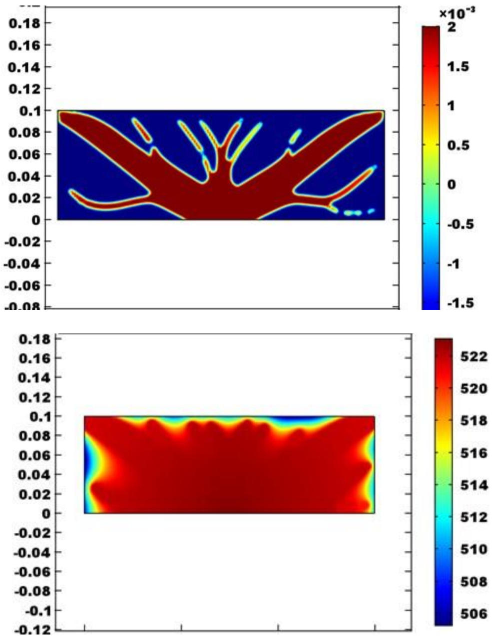

Figure 6 Level set and Temperature (K) distribution

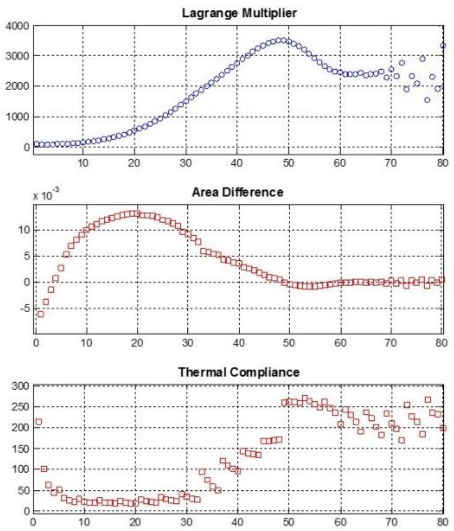

Figure 7 Convergence History

5.2 Heat sink of lower solid conductivity k_f/k_s=0.1 case:

The optimized shape and the temperature distribution within the design domain are shown in Figure-8. Unlike the high solid conductivity case, this doesn't have many branches but the primary branch connects the heat flux with the corners of the design domain. The objective and Maximum temperature observed in the design domain are given in Table-3.

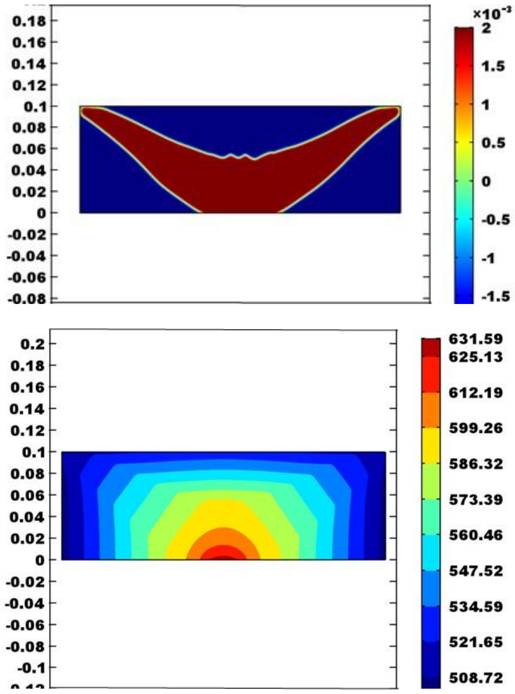

Figure 8 Optimized shape for thermal compliance (K_f/k_s=0.1) and Temperature (K) distribution

Kf/Ks	Thermal Compliance(WK/m)	Maximum Temperature (K)
0.001	202.507	523.10
0.1	3154.40	631.60

Table 3 Results of minimum thermal compliance objective optimisations

5.3 Combined TC and VD

For this case, the objective for the topology optimisation problem is defined as,

$$\text{Objective} = F1 * TC + F2 * VD \qquad (19)$$

Where F1 and F2 are factors and when F1 is equal to zero, the optimisation becomes pure VD minimisation problem and when F2 is zero it becomes TC minimisation problem.

Before optimising the heat sink for combined thermal compliance and viscous dissipation, an optimisation is carried out for pure VD minimisation case. The optimised shape and the velocity field in the design domain are shown in Figure -9.

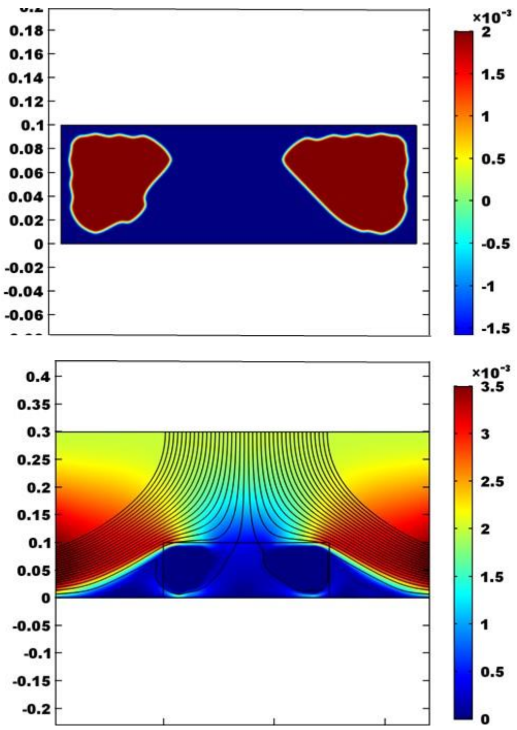

Figure 9 Optimised shape for minimum viscous dissipation and Velocity (m/s) contour

The VD magnitude is many order lower than TC, as the fluid viscosity (1.02e-3Pa.s) and Re are low. Hence for combined objectives run, F1 is taken as 1e-9 and F2 as 1, so that both the objectives will influence the optimization. Optimized shape obtained is shown in Figure-10. In order to allow the smooth flow passage, branched structure has changed into a rectangular block on top of heat source and two islet of solid region acting like a guide vane for the incoming flow. Optimized shape along with velocity and Temperature contour are given Figure-10.

(F1,F2)	Thermal Compliance (WK/m)	Viscous Dissipation (N/s)
(0, 1)	-	7.9642e-8
(1e-9,1)	2357.12	8.8307e-8

Table 4 Combined TC and VD optimisation results

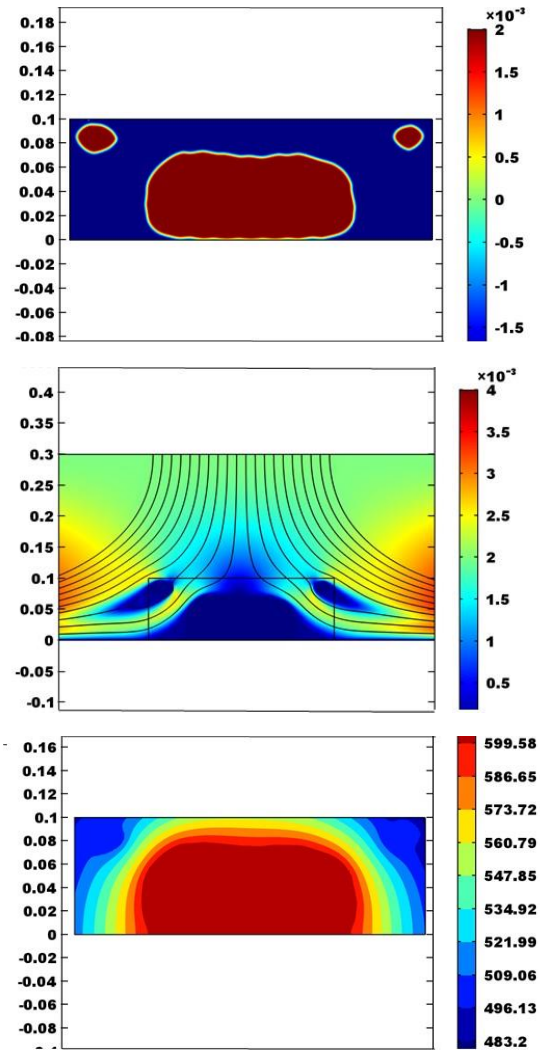

Figure 10 Results of combined TC and VD optimisation, shape, velocity (m/s) and Temperature (K) contour

5.4 Three dimensional Heat sink

A three dimensional heat sink is optimised for TC objective for conductivity ratio of k_f/k_s=0.001 subjected to laminar forced convection of Re=8. Heat sink shape, velocity and temperature distribution are given in Figure-11, 12.

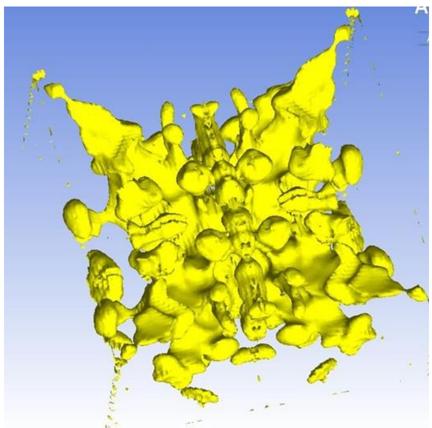

Figure 11 Top view of full (symmetrised) optimised heat sink

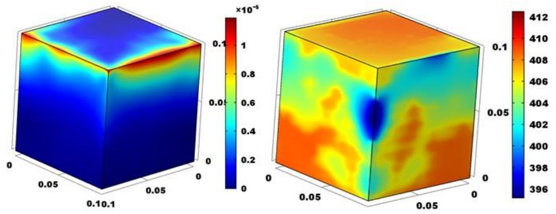

Figure 12 Velocity (m/s) and Temperature (K) distribution in the design domain

5.5 Discussion

The heat sink shapes obtained for higher solid conductivity case agrees with the tree like shape reported in literature. Because of the reduced thermal conductivity, branches are missing in the low solid conductivity case, which is understandable. Similarly for the combined TC and VD minimisation case, the optimiser tried to avoid sharp changes in temperature and velocity throughout the domain.

The 3D heat sink has a primary branch extending to the opposite diagonal and other small branches extend in different directions and the branch tip ends with bulb like mass. The structure also has isolated/disconnected regions. The symmetry boundary condition used during optimisation needs validation as it has not worked well for 2D cases.

6.0 Conclusions

Level set based topology optimisation is applied to the design of 2D and 3D convectively cooled heat sinks for different material sets. In this formulation, evolution and re-initialization of level set is carried out in MATLAB Livelink while physics is solved in COMSOL Multiphysics. This formulation ensure crisp boundary capture and no-slip condition also applied on the solid boundaries. Heat sink shape obtained for 2D and 3D higher solid conductivity case agree with tree like/dendritic shape.

The symmetry boundary condition used during 3D optimisation needs validation, this will be taken as future work. Also, extending this technique to industrial Reynolds number and for various other application is also planned in future.

7.0 References

1. Bendsoe MP, Kikuchi N, Generating optimal topologies in structural design using a homogenization method. *Comput Methods Appl Mech Eng* **71(2)**:197–224 (1988)
2. Sigmund, O. and Maute, K., Topology optimisation approaches: A comparative review, *Structural and Multidisciplinary Optimisation*, **vol 48**, pp. 1031-1055 (2013)
3. Allaire G, Jouve F, Toader A-M, Structural optimization using sensitivity analysis and a level-set method. *J Computational Phys* **194(1):**363–393 (2004)
4. Wang MY, Wang X, Guo D. A level set method for structural topology optimization. *Computer Methods in Applied Mechanics and Engineering*, **192**:227–246 (2003)
5. Challis V, Guest JK, Level set topology optimization of fluids in Stokes flow. *Int J Numerical Methods Engg* **79(10)**:1284–1308 (2009)
6. TOPLSM199 Matlab code written by MY Wang et.al: http://www2.acae.cuhk.edu.hk/ ~cmdl/download.htm
7. Z. Liu, J.G. Korvink, R. Huang, Structure topology optimization: fully coupled level set method via FEMLAB, *Structural and Multidisciplinary Optimization*, **Vol 29**, pp. 407–217 (2005)
8. Kawamoto, T.Matsumori, T.Nomura, T.Kondoh, S. Yamasaki and S. Nishiwaki, Topology optimization by a time dependent diffusion equation, *Int Jrnal for Numerical methods in Engineering*, **93:**795-817 (2013)
9. Yongbo Deng, Z Liu, J Wu, Y Wu, Topology optimization of steady Navier Stokes flow with body force, *Computer methods Applied Mechanical Engineering*, **255**, 306-321 (2013)
10. Yoon GH. Topological design of heat dissipating structure with forced convective heat transfer. *Journal of Mechanical Science and Technology*, **24(6)**:1225–1233 (2010)
11. Joe Alexandersen, Niels Aage, Casper Schousboe Andreasen and Ole Sigmund, Topology optimization for natural convection problem, *Int. J. for numerical methods in fluids*, **00**:1-23 (2013)
12. Kentaro Yaji, Takayuki Yamada, Seiji Kubo, Shinji Nishiwaki, A topology optimization method for a coupled thermal-fluid problem using level set boundary expressions, *Int Jrnal of Heat and Mass transfer* **81,**pp878-888, (2015)
13. Peter Coffin, Kurt Maute, Level set topology optimization of cooling and heating devices using a simplified convection model, *Structural and multidisciplinary optimization*, **53(5)**, pp985-1003 (2016)

8.0 Acknowledgements

The author would like to express gratitude for financial support from the University of Greenwich for sponsoring this work as part of a PhD study.