World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:9, No:4, 2015

The Effect of CPU Location in Total Immersion of Microelectronics

A. Almaneea, N. Kapur, J. L. Summers, H. M. Thompson

***Abstract*—**Meeting the growth in demand for digital services such as social media, telecommunications, and business and cloud services requires large scale data centres, which has led to an increase in their end use energy demand. Generally, over 30% of data centre power is consumed by the necessary cooling overhead. Thus energy can be reduced by improving the cooling efficiency. Air and liquid can both be used as cooling media for the data centre. Traditional data centre cooling systems use air, however liquid is recognised as a promising method that can handle the more densely packed data centres. Liquid cooling can be classified into three methods; rack heat exchanger, on-chip heat exchanger and full immersion of the microelectronics. This study quantifies the improvements of heat transfer specifically for the case of immersed microelectronics by varying the CPU and heat sink location. Immersion of the server is achieved by filling the gap between the microelectronics and a water jacket with a dielectric liquid which convects the heat from the CPU to the water jacket on the opposite side. Heat transfer is governed by two physical mechanisms, which is natural convection for the fixed enclosure filled with dielectric liquid and forced convection for the water that is pumped through the water jacket. The model in this study is validated with published numerical and experimental work and shows good agreement with previous work. The results show that the heat transfer performance and Nusselt number (Nu) is improved by 89% by placing the CPU and heat sink on the bottom of the microelectronics enclosure.

***Keywords*—**CPU location, data centre cooling, heat sink in enclosures, Immersed microelectronics, turbulent natural convection in enclosures.

I. Introduction

THE wide spread growth of data centers in recent years is resulting in an increase in end use energy demand of which more than 30% is required for their cooling [1]. Therefore more efficient cooling methods are essential, but most data centres use air cooling techniques. However, approaches that use liquids offer greater efficiency of heat removal due to their density and are readily adopted in situations of high density Datacom equipment. The revival from the 1960s [2] of liquid-cooled datacom equipment brings water from the edge of the data centre to the racks or brings water into the rack to enable liquids to get closer to the main sources of heat dissipation. There are different types of liquid cooled microelectronics which bring water into the rack or via water jackets in contact with a dielectrically immersed microelectronics enclosure.

Heat transfer in total immersion of microelectronics is achieved by natural convection of the dielectric liquid, where the heat is transferred from all of the microelectronic components, including the CPUs, to the water jacket on the opposing surface. This study develops a theoretical method for quantifying the improvements of heat transfer when the heat sink and CPU vertical location is varied as shown in Fig. 1.

The analysis presented in this paper is based on natural convection of liquid in an enclosed cavity, where there is heat flux into and out of the cavity on each vertical wall; heat is transferred by natural convection, without the use of pumps. Previously, [3] established the first theoretical benchmark for heat transfer via laminar flow in a square enclosure where the left wall is cold and the right wall is hot. Tian [4] performed benchmark experiments for the turbulent flow in a square enclosure and he plot thermal counter and vector for first time in turbulent enclosure. In the case of rectangular enclosures, [5] has investigated the fluid behaviour in both cases of laminar and turbulent flows, where it was found that for aspect ratios between 1 and 40 and Prandtl numbers between 1 and 20 the fluid flow becomes turbulent when the Rayleigh number, $Ra>10^7$. For the localised heat source in an enclosure [6] numerically investigated a vertical rectangular cavity with three heat sources on one wall and the opposing wall is maintained at a cold temperature. This study determined theoretically the optimum spacing between the heat sources and reported spacing arrangements that can drop the temperature by 10%. Heat sources in enclosures have been investigated by many further studies [7]-[9]. However, the use of fins to increase surface area has been the focus of much fewer studies. Nada [10] carried out an experimental study on finned vertical rectangular enclosures. In this case horizontal fins were adopted on the hot wall side and the opposing wall was cold. This study also investigated the effect of different Rayleigh numbers (Ra) and different fin lengths and spacing. It was found that increasing the fin length increased the Nusselt number and improves the heat transfer.

In the present work a model of turbulent natural convection flow and conjugate heat transfer is used to investigate vertically placed microelectronics in a sealed enclosure filled with dielectric liquid. The model geometry and water jacket inlet temperature remain constant. The CPU heat flux and location are varied with the aim of quantifying heat transfer improvements based on heat sink and CPU location that reduce the CPU case temperature.

II. The Model Description

The geometry and boundary conditions of the liquid immersed microelectronics model is shown in Fig. 1. Heat is

A. Almaneea (corresponding author), N. Kapur, J. L. Summers and H. M. Thompson are with the Institute of Engineering Thermofluids (iTF), School of Mechanical Engineering, University of Leeds, United Kingdom (e-mail: mnaalm@leeds.ac.uk).

generated from the CPU underneath the heat sink and cooled by the water that passes through the solid block representing the water jacket at the flow rate $3x10^{-7}$ m^3/s and inlet temperature 303 K. The conjugate heat model with symmetry planes for the immersed liquid approach is simulated via COMSOL 4.3. The simulation model is validated against the experimental work for natural heat convection in [10], where a 320mm x 200mm x 40mm enclosure with fins attached to the hot wall. The buoyancy driven flow inside the enclosure is due to the temperature difference between the hot and cold walls. The average error between the simulation model presented here and the experimental work in [10] is 4.8%.

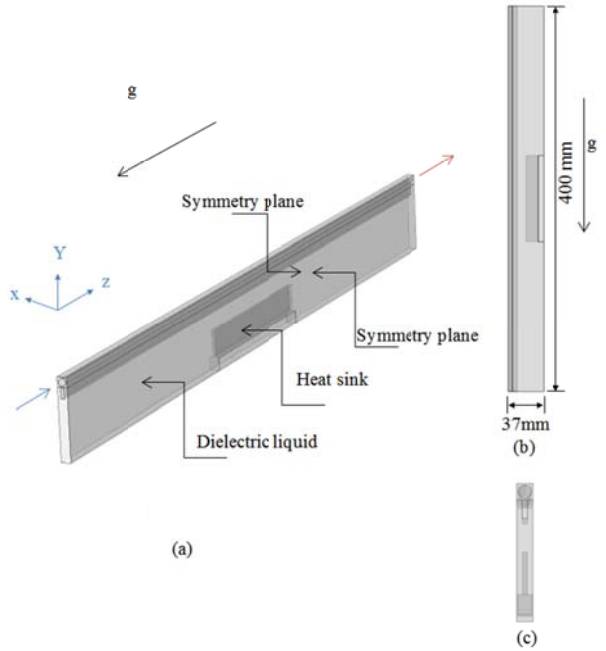

Fig. 1 The geometry of the symmetry simulation model used in COMSOL (a) Isometric view (b) Side view (c) Front View

III. Mathematical Model

In this study, the model consists of two types of fluid flow, namely dielectric liquid that circulates inside the enclosure due to natural convection and the water that is pumped through channels to carry the heat away from the enclosure. Before solving the full conjugate heat model, the two fluid flows are checked to determine whether the flow is turbulent. For the water flowing in the channel this is determined by calculating the Reynolds number, Re

$$Re = \frac{\rho_w V D}{\mu_w}$$

where D is the water channel diameter ($3.5x10^{-3}$ m), V is the average flow velocity ($1.2x10^{-2}$ m/s), ρ_w (996.6 kg/m^3) and μ_w ($7.96x10^{-4}$ Pa.s) are the density and viscosity of water, respectively. The Re is 53 for a flow rate of $1.15x10^{-7}$ m^3/s and can therefore be considered as laminar [11].

For natural convection inside the sealed enclosure, the dielectric fluid flow behaviour can be indicated by considering the Rayleigh number [12], Ra

$$Ra = \frac{g\beta\,(T_h - T_c)H^3\rho_D^2 c_p}{\lambda_D \mu_D}$$

where H is the enclosure height, g is the acceleration due to gravity, c_p is specific heat capacity, ρ_D is dielectric density, λ_D is the dielectric thermal conductivity, μ_D is the dielectric viscosity, β is the dielectric's coefficient of volumetric expansion. All these fluid thermal properties are taken from Chi et al [13] and listed in Table I. The Ra is equal to $1.8x10^8$, which is greater than 10^7 and indicates therefore that the flow is turbulent [5].

TABLE I
Working Fluid Thermal Properties

Properties	Abbrev.	Dielectric liquid
Specific heat capacity	C_p	1140 (J/kg.K)
Thermal expansion	B	$1.151496x10^{-3}$ (K^{-1})
Dynamic viscosity	μ_D	$1.124782x10^{-3}$ (Pa.s)
Thermal conductivity	λ_D	$6.9x10^{-2}$ (W/m.K)

Inside the enclosure the fluid flow is turbulent and governed by the conservation equations for mass, momentum and energy for turbulent natural convection of the dielectric liquid domain.

Continuity equation

$$\nabla.(\rho \mathbf{u_D}) = 0$$

Momentum equation

$$\rho(\mathbf{u_D}.\nabla)\mathbf{u_D} = \nabla.\left[-p\mathbf{I} + (\mu_D + \mu_{DT})(\nabla\mathbf{u_D} + (\nabla\mathbf{u_D})^T) - \frac{2}{3}(\mu_D + \mu_{DT})(\nabla.\mathbf{u_D})\mathbf{I} - \frac{2}{3}\rho k\mathbf{I}\right] + \mathbf{F}$$

The body force, **F**, depends mainly on the density variation. For an increase in temperature there is a decrease in density which creates the driving buoyancy force, expressed as:

$$\mathbf{F} = \rho(T)\mathbf{g}$$

where $\mathbf{g} = (0, 0, \alpha)$.

The dielectric fluid has density-temperature variations

$$\rho(T) = 1716.2 - 2.2T$$

In turbulent natural convection in enclosures, the k-ω turbulent model has been found to be a robust model which can offer solutions that are close to experimental observations as investigated in [14]-[16]. The k-ω turbulent model introduces two additional variables; turbulent kinetic energy, k, and specific dissipation rate, ω. The two additional transport equations are based on [17], which are:

$$\rho(\mathbf{u_D}.\nabla)k = \nabla.[(\mu_D + \mu_{DT}\sigma^*)\nabla k] + p_k - \rho\beta^* k\omega$$

$$\rho(\mathbf{u_D}.\nabla)\omega = \nabla.[(\mu_D + \mu_{DT}\sigma_\omega)\nabla\omega] + \alpha\frac{\omega}{k}p_k - \rho\beta_0\omega^2$$

The turbulent viscosity can be defined as

$$\mu_{DT} = \rho \frac{k}{\omega}$$

The production term is found from the fluid velocity as

$$p_k = \mu_{DT}\left[\nabla \mathbf{u_D} : (\nabla \mathbf{u_D} + (\nabla \mathbf{u_D})^T) - \frac{2}{3}(\nabla . \mathbf{u_D})^2\right] - \frac{2}{3}\rho k \nabla . \mathbf{u_D}$$

The energy equation would be written as:

$$\rho c_p \mathbf{u_D} \nabla T = \nabla . \left((\lambda_D + \frac{c_p \mu_{DT}}{Pr_T})\nabla T\right)$$

where λ_D is thermal conductivity and Pr_T is a turbulent Prandtl number based on the Kays-Crawford equation [17].

The empirical turbulent model constants parameters are

$$\alpha = \frac{13}{25}, \sigma_k^* = \frac{1}{2}, \sigma_\omega = \frac{1}{2}, \beta_0 = \frac{9}{125}, \beta_0^* = \frac{9}{100}, k_v = 0.41, B = 5.2$$

For the pumped water channel which is a laminar fluid domain, the governing equations are:

Continuity equation

$$\nabla . \mathbf{u_w} = 0$$

Momentum equation

$$\rho(\mathbf{u_w} . \nabla)\mathbf{u_w} = \nabla . [-p_w \mathbf{I} + \mu_w(\nabla \mathbf{u_w} + (\nabla \mathbf{u_w})^T)]$$

Energy equation

$$\rho c_p \mathbf{u_w} \nabla T = \nabla . (\lambda_w \nabla T)$$

In the solid domain, only the energy equation is required

$$0 = \nabla . (\lambda_s \nabla T)$$

where, the λ_s is the thermal conductivity for the solids which use properties of copper for the heat sink and aluminium for the water jacket and its fins.

The working fluids in this paper are water and a dielectric liquid (Table I). The water is used as a cooling liquid that passes through the channels of the water jacket and the microelectronics enclosure is filled with the dielectric liquid for which the thermal properties are kept constant except for the density which is a function of temperature.

IV. Effect of CPU Location on Heat Transfer Results

The following investigation determines the effect of the CPU and heat sink location on the heat transfer within the immersed microelectronics enclosure. The CPU and heat sink locations are varied as shown in Fig. 2. The variation is from the top to the bottom of the enclosure and the CPU location is defined as (S) where the values are listed in Table II.

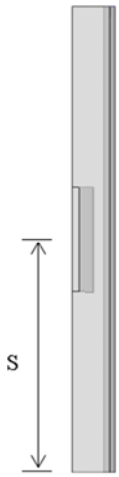

Fig. 2 CPU locations, where S is varying as shown in Table II

TABLE II
CPU LOCATION

	S (mm)
Case 1	300
Case 2	200
Case 3	100

The effect of the CPU and heat sink location on Nu is presented in Fig. 3. The heat flux load (q_L) is also varying from 70 to 100% (where heat flux q is 8888.88 W/m^2 at heat flux load q_L 100 %) while the flow rate and inlet temperature were kept constant at 3x10^{-7} m^3/s and 303 K respectively. The highest Nu implies the best heat transfer performance and it occurs when the CPU is located toward the bottom at S= 100 mm which is for case 3. Fig. 3 shows that the Nu is increased by 89% between case 1 and 3, where S= 300 and 100 mm respectively. This indicates that the CPU and heat sink located toward the bottom of the enclosure has greater effect on the heat transfer performance.

As the dielectric liquid is heated its density decreases; which drives the buoyancy induced circulation. At the top of the enclosure it is directed towards the cold plate by the presence of the top wall where it transfers heat to the cooler water jacket and then its density increases causing the dielectric liquid to travel back down to the bottom of the enclosure. This buoyancy force creates the circulation of the fluid and hence transfers the heat from the heat sink to the cold water jacket. The location of the CPU dictates the size of the flow domain over which meaningful heat transfer takes place. Fig. 4 illustrates this most clearly with the buoyancy driven eddy existing at the level of the CPU and above. The fluid below the CPU is not heated and therefore does play a role in the global flow.

When the CPU is located towards the top of the channel, a large section of the channel does not experience any flow – the recirculation sits between the CPU and the top of the wall and the fluid beneath it hardly moves. Consequently there is only a small surface area on the cool wall for removing the heat. With the CPU located at the bottom of the channel, the recirculation fills the entire cavity and the effective area over

which meaningful heat transfer takes place on the cool wall is largest – hence the temperature of the dielectric fluid will be cooler when it first meets the CPU.

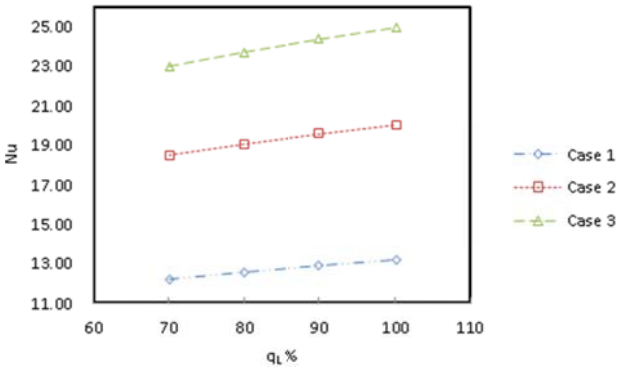

Fig. 3 Variation of heat flux load for different CPU location

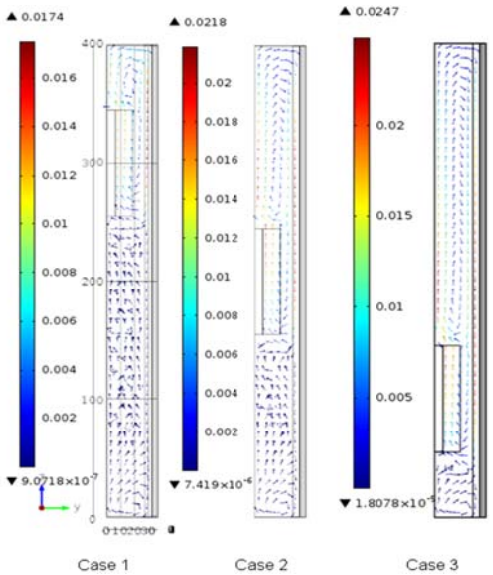

Fig. 4 Effect of the CPU location on the velocity for the three different cases. These velocity profiles are selected for the best heat transfer performance when the heat flux load is 100%, flow rate is $3x10^{-7}$ m^3/s and water inlet temperature is 303 k

V. Conclusion

This paper has investigated heat transfer of enclosed and immersed microelectronics in a dielectric liquid. Natural convection and conjugate heat transfer for a vertical enclosure is simulated using COMSOL v4.3. Since the $Ra>10^7$ in the microelectronics enclosure, the dielectric fluid regime is turbulent and the k-ω turbulence model is employed. The fluid inside the water channel is laminar because Re is very low. The model set up was compared with previous published experimental results and showed that the model in this study is in good agreement with previous work [10].

Different cases were modeled to establish the CPU and heat sink location that can yield the best heat transfer. The best CPU and heat sink arrangement was able to show a boost in the Nu number by 89%.

References

[1] Shah, A., et al. Impact of rack-level compaction on the data center cooling ensemble. in Thermal and Thermomechanical Phenomena in Electronic Systems, 2008. ITHERM 2008. 11th Intersociety Conference on. 2008: IEEE.

[2] Anderson, D., F. Sparacio, and R. M. Tomasulo, The IBM System/360 model 91: Machine philosophy and instruction-handling. IBM Journal of Research and Development, 1967. 11(1): p. 8-24.

[3] de Vahl Davis, G., Natural convection of air in a square cavity: a bench mark numerical solution. International Journal for Numerical Methods in Fluids, 1983. 3(3): p. 249-264.

[4] Tian, Y. and T. Karayiannis, Low turbulence natural convection in an air filled square cavity: part I: the thermal and fluid flow fields. International Journal of Heat and Mass Transfer, 2000. 43(6): p. 849-866.

[5] MacGregor, R., Free convection through vertical plane layers-moderate and high Prandtl number fluids. Trans. ASME, Journal of Heat Transfer, 1969. 91: p. 391-403.

[6] Phan-Thien, Y.L., Nhan, An optimum spacing problem for three chips mounted on a vertical substrate in an enclosure. Numerical Heat Transfer: Part A: Applications, 2000. 37(6): p. 613-630.

[7] Heindel, T., S. Ramadhyani, and F. Incropera, Conjugate natural convection from an array of protruding heat sources. Numerical Heat Transfer, Part A Applications, 1996. 29(1): p. 1-18.

[8] Keyhani, M., L. Chen, and D. Pitts, The aspect ratio effect on natural convection in an enclosure with protruding heat sources. Journal of Heat Transfer (Transactions of the ASME (American Society of Mechanical Engineers), Series C);(United States), 1991. 113(4).

[9] Wroblewski, D. and Y. Joshi, Liquid immersion cooling of a substrate-mounted protrusion in a three-dimensional enclosure: the effects of geometry and boundary conditions. Journal of heat transfer, 1994. 116(1): p. 112-119.

[10] Nada, S., Natural convection heat transfer in horizontal and vertical closed narrow enclosures with heated rectangular finned base plate. International Journal of Heat and Mass Transfer, 2007. 50(3): p. 667-679.

[11] Çengel, Y.A., R.H. Turner, and J.M. Cimbala, Fundamentals of thermal-fluid sciences 2008. p. 534.

[12] Holman, J., Heat transfer, 9th. 2002, McGraw-Hill. p. 335-337.

[13] Chi, Y.Q., Jonathan Summers, Peter Hopton, Keith Deakin, Alan Real, Nik Kapur and Harvey Thompson, Case Study of a Data Centre Using Enclosed, Immersed, Direct Liquid-Cooled Server, in Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM). 2014, IEEE.

[14] Zitzmann, T., et al. Simulation of steady-state natural convection using CFD. in Proc. of the 9th International IBPSA Conference Building Simulation 2005. 2005: Montréal: IBPSA.

[15] Rundle, C. and M. Lightstone. Validation of turbulent natural convection in a square cavity for application of CFD modelling to heat transfer and fluid flow in atria geometries. in 2nd Canadian Solar Buildings Conference, Calgary. 2007.

[16] Aounallah, M., et al., Numerical investigation of turbulent natural convection in an inclined square cavity with a hot wavy wall. International Journal of Heat and Mass Transfer, 2007. 50(9): p. 1683-1693.

[17] Wilcox, D.C., Turbulence modeling for CFD. Vol. 2. 1998: DCW industries La Canada, CA.