Natural Convection Heat Loss from A Partly Open Cubic Enclosure

Timothy N Anderson[1,a *] and Stuart E Norris[2,b]

[1]Auckland University of Technology, New Zealand

[2]University of Auckland, New Zealand

[a]timothy.anderson@aut.ac.nz, [b]s.norris@auckland.ac.nz

*Corresponding author

Keywords: natural convection, open enclosure

Abstract. Natural convection heat transfer in enclosures is an area that has particular significance to a wide range of applications. However, heat transfer from enclosures with openings to the surroundings, such as open windows in buildings or passively ventilated electronics enclosures, have received far less attention. In this regard, there is still a lack of generalised relationships that can be used in determining the heat transfer in partially open enclosures. As such, this work presents an experimental and computational investigation of the natural convection heat loss from partly open cubical enclosures, with various opening configurations, with a view to understanding the mechanism and developing relationships that describe it. It shows that heat transfer from the partly open enclosures is most strongly influenced by the Rayleigh number and the opening size.

Introduction

Natural convection heat transfer from enclosures with openings to the surroundings is an area of work that has particular significance to the development of energy efficient buildings. Despite this, there are few studies that have examined the issue of heat loss from open or partly open enclosures.

The majority of the work undertaken on open enclosures has examined the issue of natural convection as it relates to ventilation rates in buildings, as well as more fundamental studies of the convection mechanism. One such example is the work of [1] who computationally examined the flow in a square cavity with multiple slotted openings. They found that the Nusselt number and the volumetric flow rate both increased with Rayleigh number and also the opening ratio.

In their work [2] computationally examined buoyancy affected flows in a room like enclosure with ventilation and noted the flow was affected by the geometry. Similarly, [3] examined the ventilation of buildings through large openings on a single side and with sun shades. They found that the airflow could be related to the Archimedes number.

In a more fundamental sense, [4] examined buoyancy driven ventilation of a room with a heated floor. They performed a series of experiments of the flow in their enclosure, and then developed a model of the flow under generalised conditions. They noted that there is a need for further work in the field with a particular emphasis on the opening size.

Despite the insights of the studies on ventilation rates, there is also a need to relate these to the heat transfer. In their study [5] undertook a computational fluid dynamics (CFD) analysis of the heat transfer by laminar natural convection from electronic components housed in a vented enclosure. They found that both location and size of the openings influenced the flow and temperature in their enclosure. Further, studies such as the work of [6] and [7] have explored the influence of opening aspect ratio and inclination on the heat transfer from open solar receivers. As such, they have delivered a number of relationships to describe heat transfer as a function of these two parameters as well as the Rayleigh or Grashof number.

Despite the work that has been undertaken, there is still a lack of relationships that can be used in determining the heat transfer in partially open enclosures. In the work of [8] the effect that varying convective heat transfer coefficients in building energy simulation models is discussed; in this it is shown that varying the value of this only slightly can significantly affect the predicted heating energy required in buildings. Therefore, with partly open buildings, this prediction becomes even more difficult, and so there is a need to develop relationships to describe the impact this has on convective heat transfer coefficients. As such, this work aims to develop an understanding of this problem with a view to developing such a relationship.

Numerical Modelling

As an initial step to determining the natural convection heat transfer in a partially open enclosure, a steady state model was formulated in a commercial CFD solver [9] based on the finite volume method. The solver utilises the Reynolds averaged Navier-Stokes equations in the prediction of turbulent flows and a Cartesian coordinate system to spatially distribute a rectangular computational mesh [9]. In its treatment of turbulence, the solver employs transport equations for the turbulent kinetic energy and turbulence dissipation rate using the standard k-ε model. Additionally, it uses a Modified Wall Function approach, where a Van Driest's profile is utilised to describe the near wall flow. It has been suggested that this treatment provides accurate velocity and temperature boundary conditions in the conservation equations [9].

The solver utilises a cell-centred approach to obtaining a conservative approximation of the governing equations. The second-order upwind approximations for the fluxes are treated using the QUICK scheme and a Total Variation Diminishing (TVD) method. Finally, a SIMPLE-like approach and an operator-splitting technique are used in the treatment of time-implicit approximations made in the continuity and convection/diffusion equations. As such, the solver is able to provide accurate prediction of natural convection in a square enclosure when compared with the benchmark numerical solution of [10].

For this study a 3-dimensional air filled cubic enclosure with single side openings of varying open aspect ratio was examined, as shown in Fig. 1. The simulation was conducted in a much larger computational domain (to limit the impact of using an ambient/open boundary condition [9]) which, after a mesh dependence study, consisted of a mesh of approximately 500,000 cells. For the simulations, the floor of the enclosure was heated at a constant temperature while all other walls were assumed to be adiabatic, this corresponded to Rayleigh numbers in the range $1x10^8$ to $5x10^8$, taking the enclosure height to be the characteristic length. In this regard the aim was to approximate the effect of an opening on the heat loss from a room where solar radiation had been shining on the floor, thus leading to an increase in its temperature.

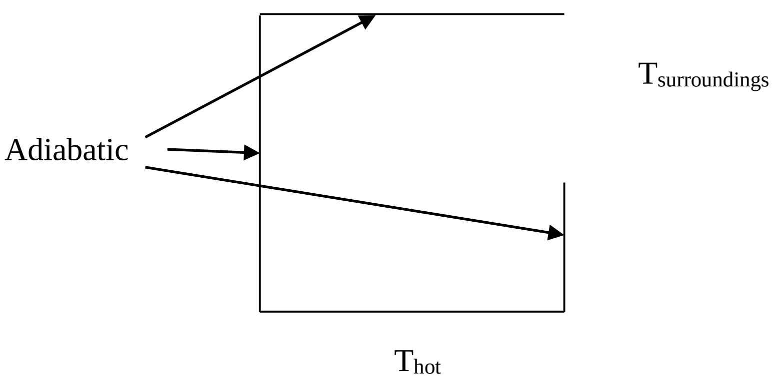


Fig. 1 Schematic of the computational boundary conditions

Numerical Results

In considering the results of the simulations, it was apparent that increasing the temperature of the heated floor led to an increase in heat transfer from this surface. It was noticeable that as the opening size increases, so does the heat transfer from the heated wall as shown in Fig. 2 and also by [6] and [7]. In this regard the Nusselt number for the heated wall could be represented as a function of the opening aspect ratio (d/D) and the Rayleigh number.

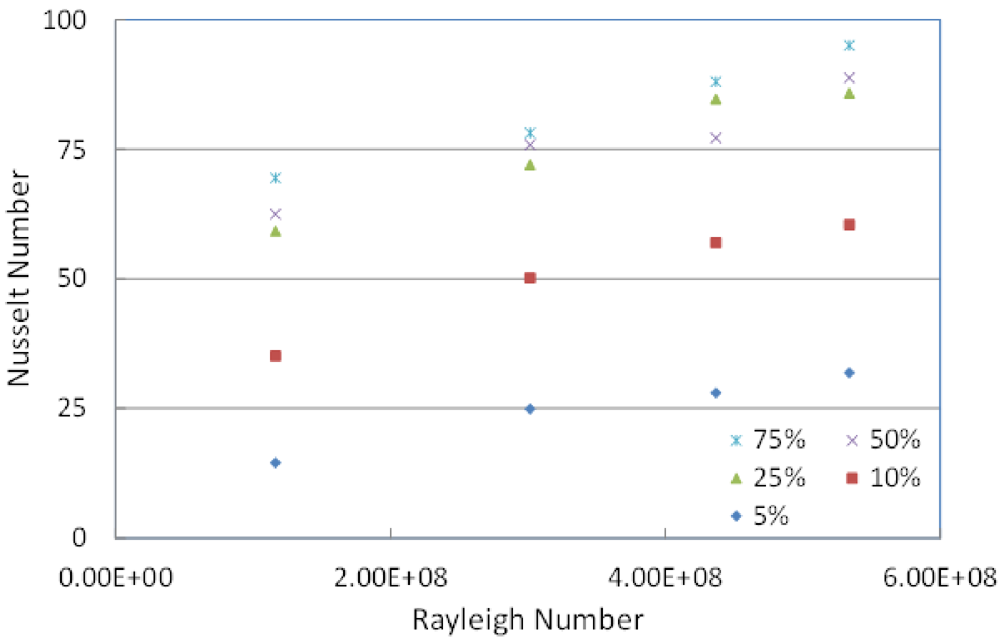


Fig. 2 Nusselt v Rayleigh Number for an open enclosure with various opening sizes

When considering the convective heat loss from the enclosure, it is interesting to examine the flow in the enclosure. In Fig. 3 it can be seen that for a small opening, a significant part of the flow is recirculating in the enclosure whereas for a larger opening there is a greater exchange of flow between the enclosure and the surroundings. In addition, the isotherms show a plume of cold air extends almost the entire height of the enclosure for large openings, leading to a large temperature gradient near the enclosure floor; these factors would explain why the heat transfer is higher with larger openings.

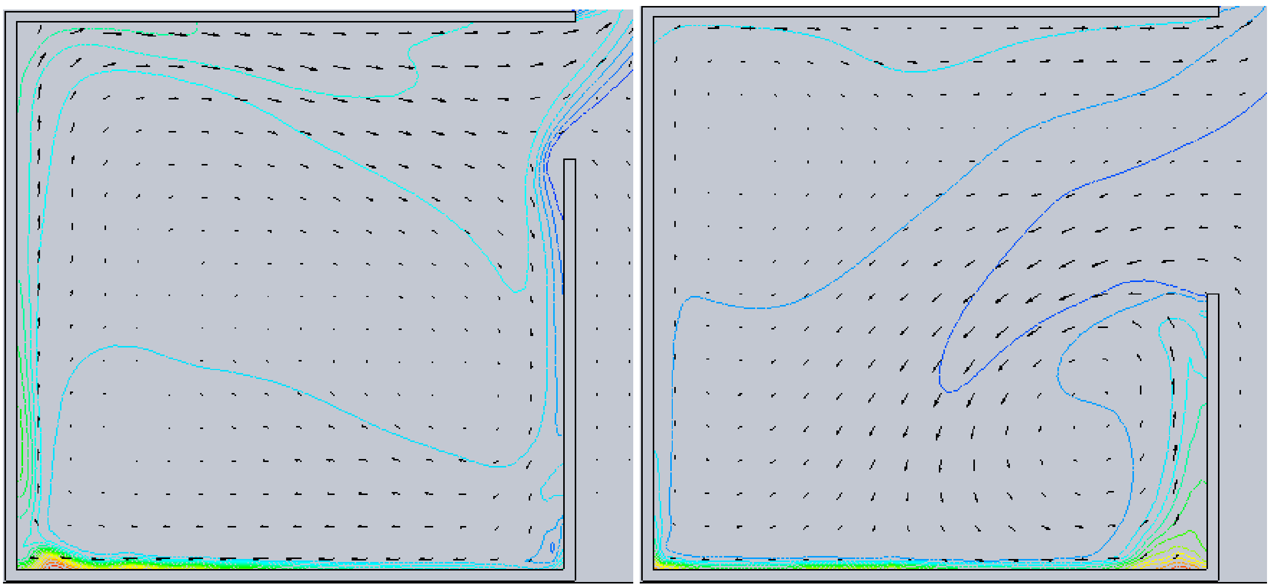


Fig. 3 Flow in an open enclosure with various opening sizes (25% left, 50% right)

Experimental Setup

The computational modelling of the partly open enclosure suggested that the Rayleigh number and also the size of the opening played an important role in the determination of the flow and heat transfer for such systems. In order to verify this observation, it was decided to develop an experiment to explore this further.

As such, an acrylic tank 1.2m x 1.2m x 0.6m was designed to contain a small partly open acrylic enclosure that was subsequently submerged under water (rather than in air). The acrylic open cavity was a cube with internal dimensions of 100mm and a wall thickness of 4.5mm. The cube cavity had one vertical side open to the surrounding fluid in the larger tank, and a set of walls of different heights were designed to fit into the vertical open wall. In this work two locations were tested for the opening; top open and bottom open.

As with the computational model the floor of the small enclosure was heated. This was achieved using a stick-on resistance heating element attached to the back of a 5 mm thick aluminium plate in order to provide a uniform surface temperature at the top surface. The heater plate had four T-type thermocouples uniformly distributed across its surface to record the temperature. Mineral wool was packed under the heating element to eliminate heat loses through the bottom. In this manner the heat transfer was predominantly from the surface of the aluminium plate in contact with the water in the tank. In addition, three thermocouples where then used to measure the fluid temperature in the tank (thus giving the temperature difference between the heater and the surroundings). To vary the Rayleigh number in the enclosure, the temperature difference was modified using a variable DC power supply connected to the resistance heating element. The power being drawn by the heater was controlled by varying the voltage with a fixed current. To determine the temperature gradient in the enclosure (between the heater and surrounding water) the thermocouples were connected to a Picolog TC-08 eight channel thermocouple data acquisition system and recorded by a computer via the USB interface.

To characterise the flow, the water was seeded with neutrally buoyant glass microspheres to allow a PIV analysis to be undertaken when the system reached steady state thermal conditions. The flow was illuminated by a 1W diode pumped solid state laser generating a light sheet approximately 1mm thick. A series of 1000 consecutive images of the flow were captured by a 5 megapixel CCD camera, with an inter-frame time of 60ms, and used to determine the time-averaged flow inside the enclosure using the MicroVec PIV processing package [11]. Fig. 4 shows a schematic representation of the experimental setup.

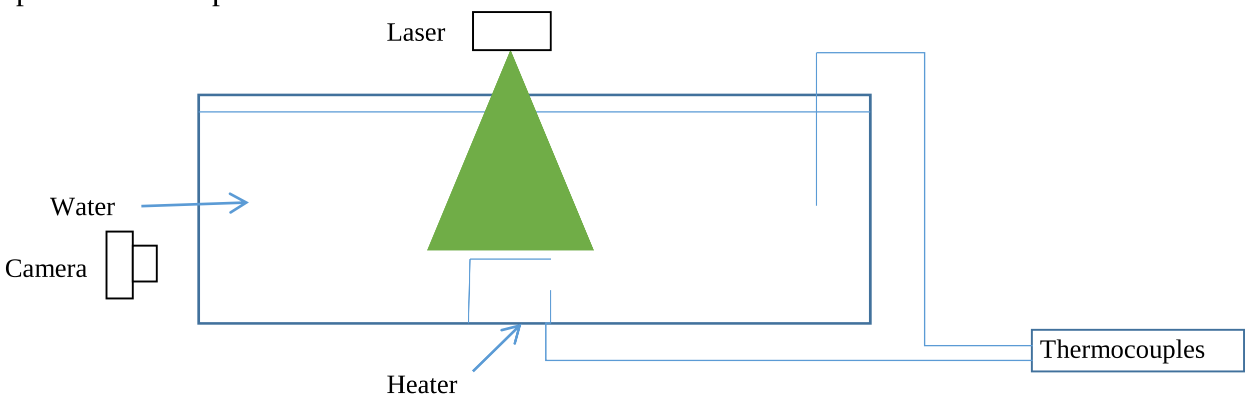
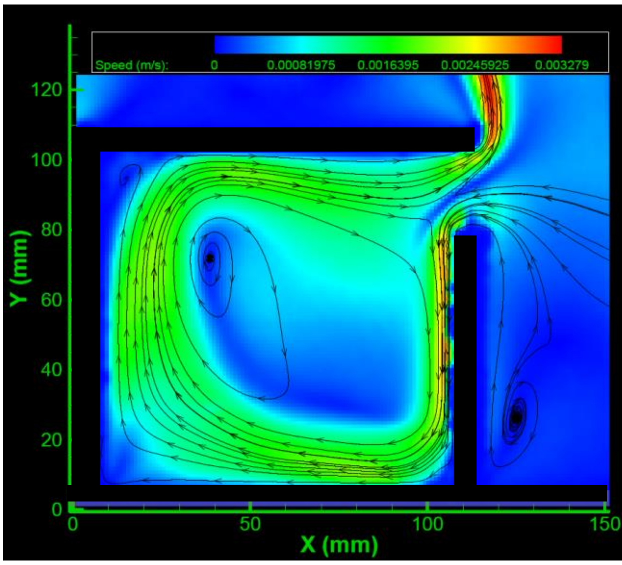


Fig. 4 Schematic of experimental apparatus

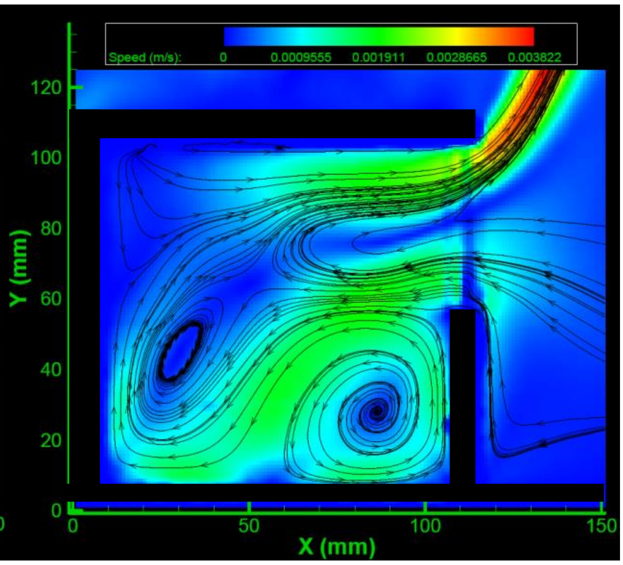
Experimental Results

In Fig. 5a-d it can be seen that there is a distinct neutral axis for the openings, representing the transition from inward to outward flow, at approximately 50% of the opening height. However, more importantly, it can be seen that the opening size has a dramatic influence on the flow structure within the enclosure, as had been elucidated by the CFD simulations. In Fig. 5a-c with a top opening, the flow initially resembles that of a closed cube (25% open), as this increases to 50% two strong circulating cells develop due to the large plume entering the enclosure while at 75% open these cells diminish with only a small cell occurring behind the low wall, as has been observed in the simulation results.

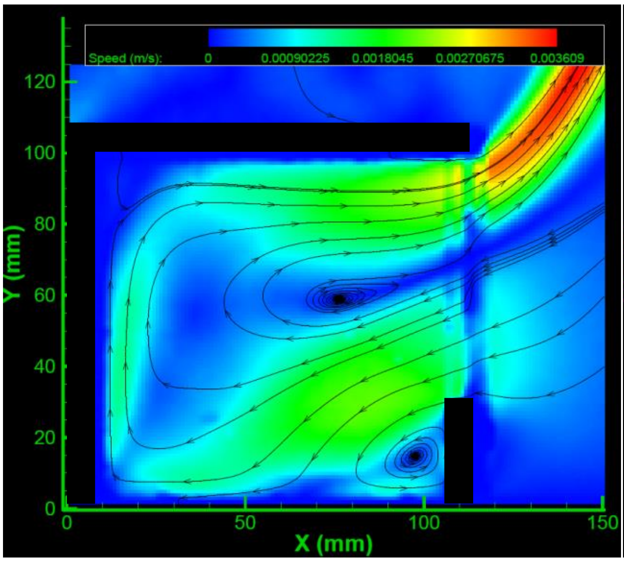
What is more interesting is the difference between Fig. 5c-d, both of which are 75% open but with a different opening location. Despite the opening being the same size, there are significant differences in the flow structure as a result of the orientation. In this respect, there may be differences in the local heat transfer coefficient, or temperature, across the heated floor due to the velocity differences in this area. In turn, for partly open enclosures such as buildings with open windows, the observed flow structure, velocity variations and temperatures might affect the comfort of the occupants and the air quality. In this respect there may be an ideal location for these openings, however this is an area that would require further investigation.



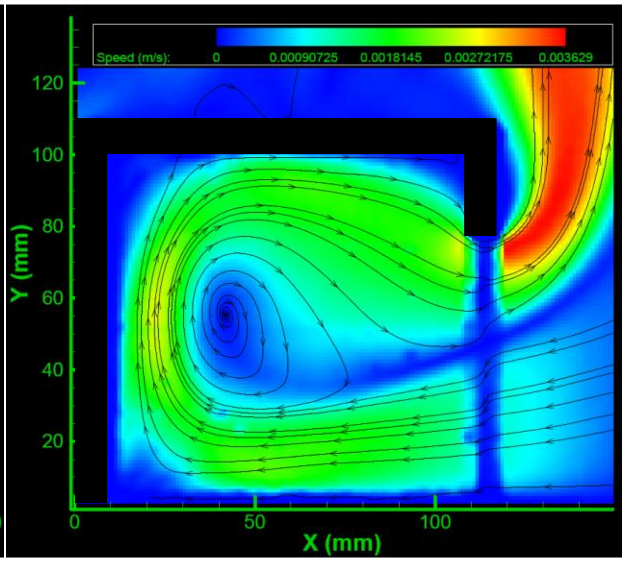
(a) 25% open at top



(b) 50% open at top



(c) 75% open at top



(d) 75% open at bottom

Fig. 5 Streamlines and velocity contours for enclosures at 200W/m^2

Discussion and Conclusion

Natural convection heat transfer from enclosures with openings to the surroundings, such as open windows or doors in buildings, has received little attention and so there is a lack of relationships that can be used in determining the heat transfer in such systems. In order to develop more accurate modelling tools that describe the thermal characteristics and indoor air quality in low energy buildings, the development of such relationships is paramount.

This work has shown both computationally, that heat transfer from partly open enclosures is strongly influenced by both the Rayleigh number and also the opening size. Such a finding, though significant requires further work to develop a generalised relationship for a wider range of Rayleigh numbers. Further, from the experimental analysis, it is possible that the location of such openings may also influence the nature of a relationship to describe this heat transfer and would also influence the local heat transfer coefficients which would be encountered from the heated base. As such, there is still a significant amount of work to be undertaken in the development of such expressions.

References

[1] E. Bilgen, A. Muftuoglu, Natural convection in an open square cavity with slots, International Communications in Heat and Mass Transfer, 35 (2008) 896-900

[2] P. V. Nielsen, A. Restivo, J. H. Whitelaw, Buoyancy-affected flows in ventilated rooms, Numerical Heat Transfer, 2(1) (1979) 115-127

[3] A.A. Argiriou, C.A. Balaras, S.P. Lykoudis, Single-sided ventilation of buildings through shaded large openings, Energy, 27(2) (2002) 93-115

[4] C. Gladstone, A.W. Woods, On buoyancy-driven natural ventilation of a room with a heated floor, Journal of Fluid Mechanics, 441 (2001) 293-314

[5] E. Yu, T. Joshi, A numerical study of three-dimensional laminar natural convection in a vented enclosure, International Journal of Heat and Fluid Flow, 18 (1997) 600-612

[6] M. Prakash, S.B. Kedare, J.K. Nayak, Numerical study of natural convection loss from open cavities, International Journal of Thermal Sciences, 51 (2012) 23-30

[7] M. Miyamoto, T.H. Kuehn, R.J. Goldstein, Y. Katoh, Two-dimensional laminar natural convection heat transfer from a fully or partially open square cavity, Numerical Heat Transfer Part A: Applications, 15(4) (1989) 411-430

[8] K.J. Lomas, The U.K. applicability study: an evaluation of thermal simulation programs for passive solar house design, Building and Environment, 31(3) (1996) 197-206

[9] Cosmos, COSMOS FloWorks Fundamentals, Solidworks Corporation, Concord MA, 2006

[10] G. de Vahl Davis, Natural convection of air in a square cavity: a benchmark numerical solution, International Journal of Numerical Methods in Fluids, 3(3) (1983) 249–264

[11] MicroVec, MicroVec 3.5.1. User Manual, Beijing Microvec Inc. Beijing, 2013