

MODELLING BUOYANT THERMAL PLUMES IN NATURALLY VENTILATED BUILDINGS

Faisal Durrani¹, Malcolm J Cook², and James J McGuirk²

¹School of Civil and Building Engineering, Loughborough, LE11 3TU, UK

²Department of Aeronautical and Automotive Engineering, Loughborough, LE11 3TU, UK

ABSTRACT

The aim of the work reported in this paper was to evaluate the performance of Large Eddy Simulation (LES) for modelling natural ventilation driven by twin plumes. The flow is characterised by an interface height which separates the warm buoyant air above from the cooler air below, and a merging height for coalescence of the two plumes. Comparison between the LES predictions and theory for the interface height and volume flow rate in the merged plumes is good, giving confidence that LES has potential for modelling this important class of flows.

INTRODUCTION

Natural ventilation can be used as part of a low energy strategy in reducing carbon emissions from buildings. Buoyancy-driven natural ventilation harnesses the buoyancy forces associated with temperature differences between the interior and exterior environments to drive a ventilation flow through a building. Amongst other parameters, the behaviour of thermal plumes and especially their interaction with each other (e.g. thermal plumes from a computer and its user) contributes largely to the final ventilation flow pattern. Heat sources within naturally ventilated buildings can be represented by sources that are isolated (Abdalla et al., 2007), a line heat source (Bastiaans et al., 2000) or by a uniformly distributed heat source over the entire floor area (Gladstone and Woods, 2001). Kaye and Linden (2004) using salt bath modelling and Durrani et al. (2011) using a Reynolds averaged Navier Stokes approach, studied the interaction of twin thermal plumes.

Computational Fluid Dynamics (CFD) has been used widely to analyse natural ventilation. However, much of this work has considered steady-state solution techniques using RANS. Due to their inherent time averaging nature, RANS can significantly reduce the computational time but at the cost of losing instantaneous flow behaviour of any turbulence present in the flow. Hence, for those unsteady turbulent flow applications where the unsteady motions are of importance, Large Eddy Simulation (LES) can provide an alternative option in terms of high accuracy of simulation and acceptable

computational time. With the recent developments in high performance computing, it has become possible to use transient methods such as LES. LES resolves the large eddies which carry most of the energy of the flow and play the main part in the resulting flow patterns. The effects of the smaller scales, which are mostly isotropic, are represented using an approximate mathematical model known as a 'sub-grid scale' model. This reduces the computational power required as compared to a Direct Numerical Simulation (DNS) approach in which the entire flow field is resolved.

To the authors' knowledge, very little work has been done to study buoyancy-driven thermal plume interactions using LES. Nam and Bill Jr (1993) simulated turbulent plumes using a modified k- ϵ turbulence model. Zhou et al. (2001) and Abdalla et al. (2007) used the LES approach to study thermal plumes, however the former focused on forced plumes while the latter on single isolated plumes.

The aim of the work presented in this paper was to simulate twin thermal plumes in a naturally ventilated enclosure using LES. The results are intended to elucidate the interaction of the two plumes and to evaluate the performance of LES in modelling natural ventilation by comparing the results with existing analytical models.

EXPERIMENTAL AND ANALYTICAL WORK

Linden and Kaye (2006) suggest that for two unequal plumes the interface height is given by:

$$\xi' = \xi + \varepsilon \quad (1)$$

where ξ is the normalised interface height and ε is the small perturbation to the interface height given by:

$$\varepsilon \approx -\xi_v \left[\frac{5\xi_1^4}{\left(\frac{A^*}{H^2 C^{3/2}}\right)^2 + 5\xi_1^4} \right] \quad (2)$$

For two unequal plumes with a buoyancy flux ratio $\psi=0.5$ the location of the virtual origin ξ_v is given by Figure 9, Kaye and Linden (2004):

$$\xi_v = \frac{0.11 \times x_o}{\alpha} \quad (3)$$

where, x_o is the heat source separation and α is the entrainment coefficient.

Linden et al. (1990) derived the relationship between the normalised interface height $\xi = h/H$ and the effective opening area, A^* as:

$$\frac{A^*}{H^2} = C^{3/2} \left[\frac{\xi^2}{1 - \xi} \right] \quad (4)$$

where,

$$C = \frac{6\alpha \left[\frac{9\alpha}{10} \right]^{1/3} \pi^{2/3}}{5} \quad (5)$$

Linden et al. (1990) used a value of 0.1 for the entrainment coefficient α .

A^* is the effective opening area formulated by Hunt and Linden (2001) to represent openings of area a_t and a_b at the top and bottom of the space respectively:

$$A^* = \frac{C_D a_t a_b}{\left[\frac{1}{2} \left\{ \left(\frac{C_D^2}{C_e} \right) a_t^2 + a_b^2 \right\} \right]^{1/2}} \quad (6)$$

where C_D and C_e are the coefficients of expansion and discharge respectively.

Linden et al. (1990) utilised 'top-hat' profiles to represent their Gaussian counter-parts. Figure 1 illustrates a profile of the Gaussian and top-hat velocity profiles. The relationship between the top-hat quantities and their Gaussian counterparts for a 3D point heat source are given by Cook (1998):

$$V_T = \frac{V_G}{2} \quad (7)$$

$$b_T = \sqrt{2} b_G \quad (8)$$

The volume flux is given by:

$$Q = \pi \times b_T^2 \times V_T \quad (9)$$

and the buoyancy is calculated using:

$$F = \frac{g\beta q}{\rho C_p} \quad (10)$$

where, g is acceleration due to gravity, β is the expansion coefficient, q is the strength of the heat source, C_p is the specific heat capacity and ρ is the reference density.

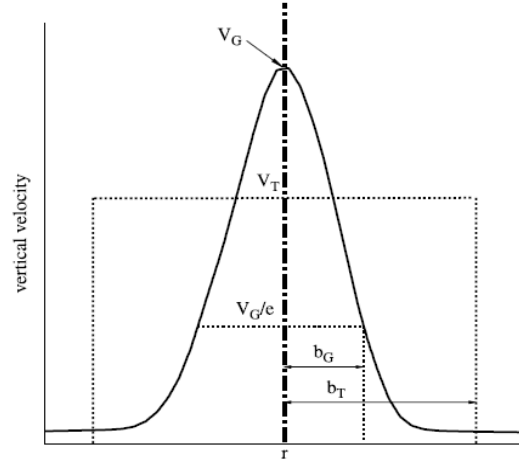


Figure 1: Velocity profile of a plume representing the Gaussian and "top-hat" profiles (source: Cook, 1998)

METHODOLOGY

The Computational Domain and Boundary conditions

The computational domain selected was a rectangular enclosure with a floor area of 1m x 1m and height 1m (Figure 2). Two heat sources, q_1 and q_2 were located close to the centre of the floor each with area 0.0009m² and separated by a distance of 0.1m (centre to centre). The strength of the heat sources was $q_1=20W$ and $q_2=10W$. Long rectangular openings were specified along the top and bottom side edges of the enclosure each with an open area of 0.05m². The two openings at the bottom of the enclosure allow ambient air into the domain while the pair of openings at the top allows warm air out of the domain.

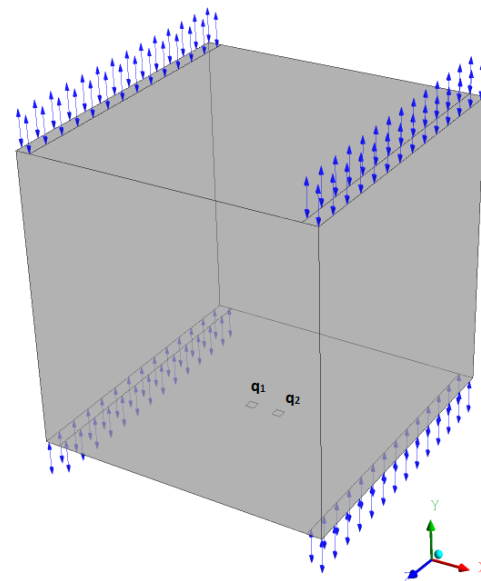


Figure 2: Computational domain in CFX

At each opening, the following condition was imposed on pressure:

$$\Delta p_{loss} = -\frac{1}{2}f\rho U_n^2 \quad (11)$$

where, f is the loss coefficient and U_n is the velocity component normal to the opening boundary.

Initial conditions were such that the velocity components in the computational domain were set to zero and the temperature inside the room was the same as the ambient. This would ensure that the plumes would evolve in a stagnant flow and that stratified flow would develop quickly.

The software package and LES method

The software package used for the CFD simulations was CFX (2011). This software uses the finite volume method to discretise the incompressible continuity, momentum and energy equations. The solver is fully implicit thus making it stable with respect to time step size. The LES technique was applied to resolve the flow with the Smagorinsky model (Lilly, 1967) as the sub-grid scale model.

The selection of the time step was based on the value of the Courant number (CFL). The CFL number reflects the distance that fluid will flow in one time step, as a proportion of a cell length and is given by:

$$CFL = \frac{V \times \Delta t}{\Delta x} \quad (12)$$

where V is the linear velocity, Δt is the time step size and Δx is filter width.

The simulation ran with an adaptive time step initially with a value of 0.01s. The adaptive time step criterion was to maintain the CFL number in the range of 0.5-1.0. Once this criterion was achieved, the simulation was continued with a constant time step time step value i.e. 0.02s.

Although the solver is implicit, for accuracy it is recommended that the CFL number is maintained in the range of 0.5-1.0. Coefficient loops solve the equations within each time step. In this work, the aim was to keep the inner coefficient loops within the 3 to 5 range. The convergence criteria for each time step were that the root mean square (RMS) residuals of velocity, temperature and pressure should be less than 1×10^{-6} .

RESULTS

Determining convergence

Steady state is reached when the flow in the computational domain has evolved and the transition and turbulent mixing is established. As per Linden et al. (1990) two homogeneous stratified layers are expected. Thus, temperature monitor plots were placed in the domain (but away from the two plumes) from the floor to the ceiling at 0.2m intervals to monitor the formation of such a regime.

A statistically steady state was considered to be achieved when the following criteria had been met:

- ventilation flow rate was unchanging;
- velocity, temperature and pressure values for multiple monitor points inside the domain were stable

These criteria were achieved after 90s of simulation time. In addition, a buffer of another 30s was modelled followed by a further 30s during which statistical analysis of the turbulence was carried out.

Mean flow field

Once the flow had reached steady state, the performance of LES was tested by comparing the interface height prediction with theory. With the formulation presented by Linden and Kaye (2006), for unequal plumes merging below the interface height, the normalized interface height should be $\xi'=0.677$.

Figure 3 illustrates the steady state mean temperature field. A vertical temperature profile plotted away from the plume centres helped determine the interface height (Figure 4).

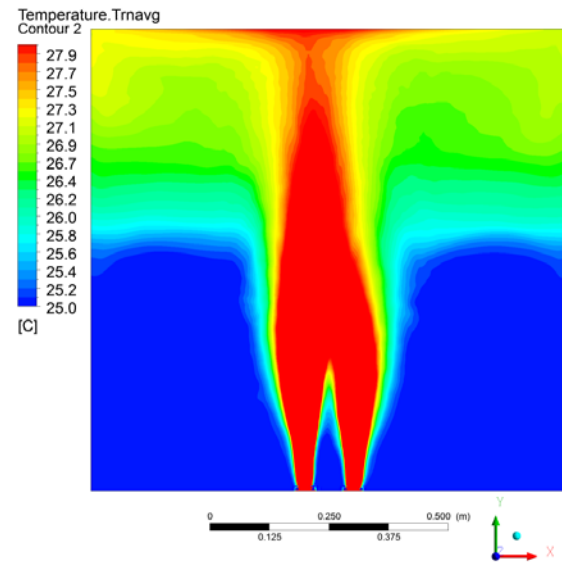


Figure 3: Mean temperature contour (x - y plane, $z=0.5m$)

As can be seen from Figure 4 the air temperature remains constant below the height of 0.5m and above 0.8m with the interface located within this range. This is in contrast to the step change in temperature at the interface height shown by the analytical models. The transition layer between the lower ambient temperature fluid and the upper warm fluid extends over approximately 0.2m. Cook (1998) and Abdalla et al (2007) have also predicted this 'smeared' interface using RANS and LES respectively. Abdalla et al (2007) suggest that this may be attributed to the turbulent diffusion of heat induced by the unsteady motion at the interface.

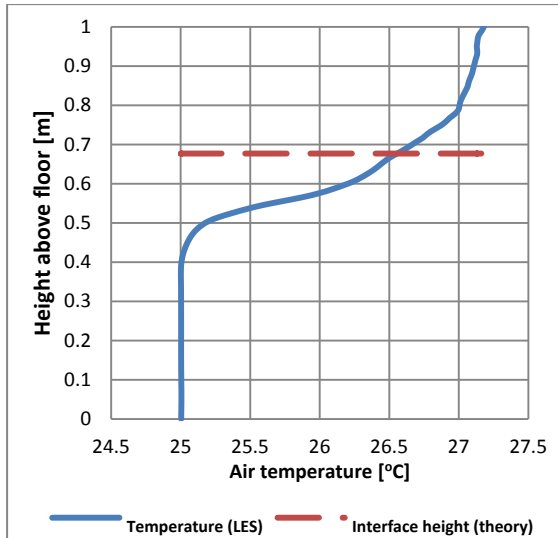


Figure 4: Mean temperature profile, $(x,y,z)=(0.1,0-1,-0.1)$

Instantaneous field

Figures 5 and 6 display the instantaneous temperature profiles passing through the centre of the domain. Figure 6 helps in visualising the 3D spread of the plume.

The plots qualitatively suggest that the plumes are turbulent for the full distance from the heat source to the ceiling. This was indicated by the presence of no laminar flow patterns. To confirm the 3D turbulent motion of the plume, a graph is plotted of the instantaneous w -velocity component along the z -axis normalized by the maximum velocity along y -axis (v_{max}). Figure 7 illustrates this three dimensional motion and suggests that both the plumes contain significant three dimensional motion right from the heat sources. In addition, it is worth noticing that the plume of q_1 is more turbulent than the plume of q_2 as indicated by the larger variations in w/v_{max} value.

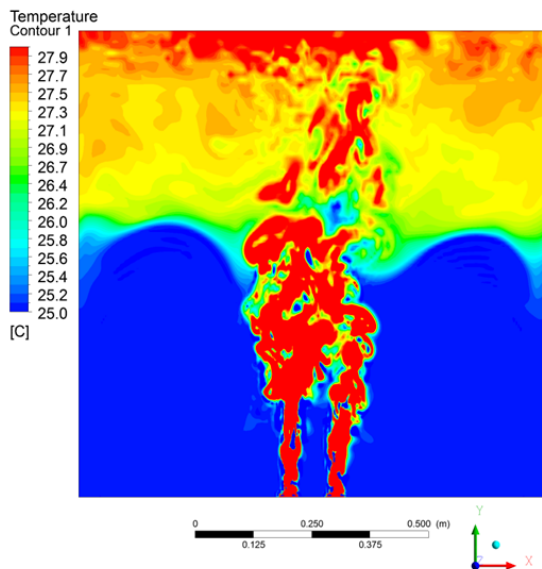


Figure 5: Temperature contour over an x - y plane $(z=0.5)$

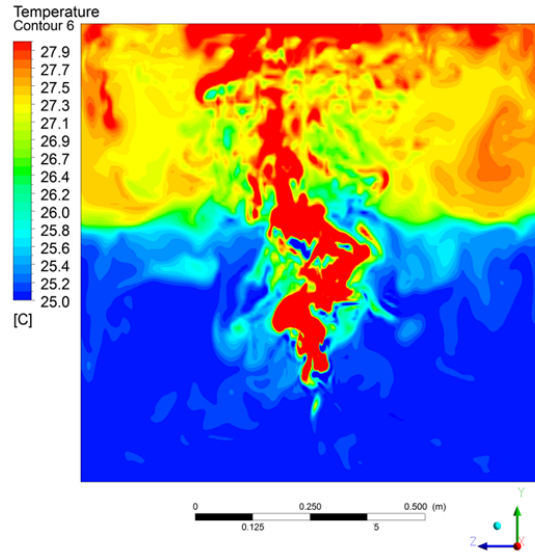


Figure 6: Temperature contour over a y - z plane $(x=0.5)$

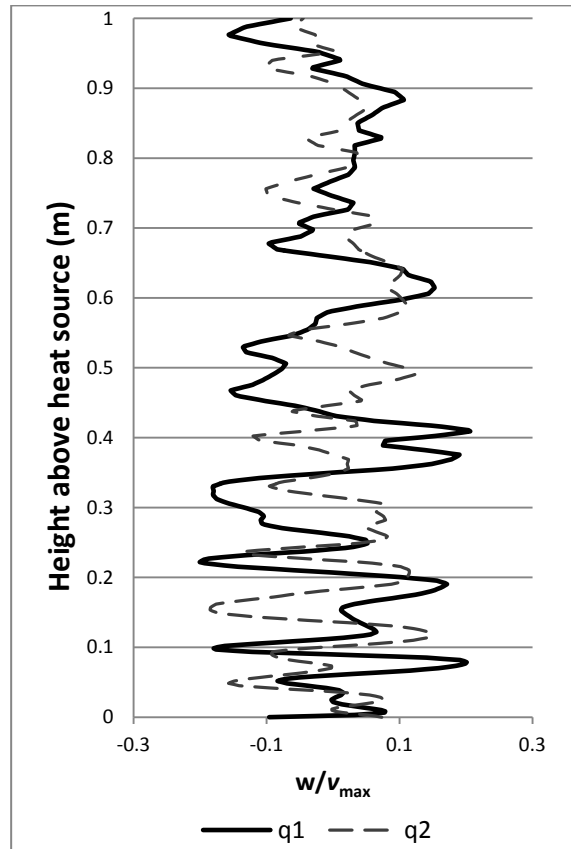


Figure 7: Instantaneous velocity along z -axis normalised by max value of velocity along y -axis

Close to the heat sources the two plumes entrain surrounding air, they then grow in a radial direction until they merge with one another. This results in a merged plume of warm buoyant air that rises up towards the ceiling after which it spreads to the sidewalls and forms the upper layer of warm air.

Figure 5 and 6 also confirm the presence of smaller coherent structures in the upper warm layer. It is worth noticing that the interface surface is not planar but rather undulating over a height of approximately 0.2m. This supports the findings in the mean flow field plot (figure 4) suggesting that the temperature across the interface is more diffuse than a step change. Small scale information that is obtained using LES such as the structure of small scales in the plumes is not available when using a RANS simulation. Figure 8 shows a RANS simulation for a similar case (Durrani et al., 2011).

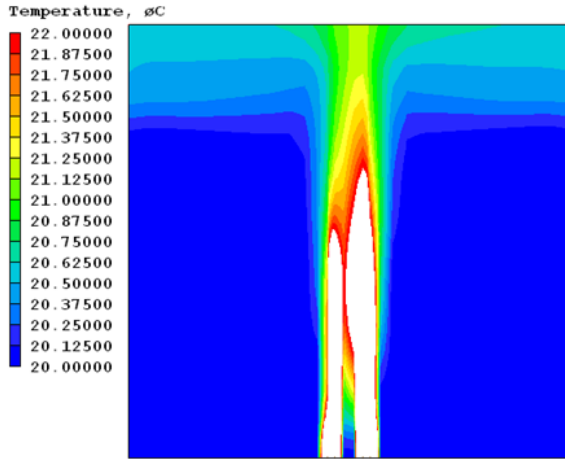


Figure 8: Temperature contour using RANS

Plume merging height

In order to determine the merge height (z_m) of the two plumes, the buoyancy or the velocity profiles can be plotted. These Gaussian profiles can be used to determine the merge height (Kaye and Linden, 2004). When the velocity profiles of the two plumes have merged it is reasonable to consider the plumes to have merged. Figure 9 illustrates the merging of the velocity profiles of the two plumes. It can be observed that above the height of 0.55m the profiles coalesce and hence the plumes are considered to have merged. It was also noted that the ratio of the peak velocities was $\psi^{1/3}$ whereas the ratio of the peak buoyancies was $\psi^{2/3}$ which was in accordance with Kaye and Linden (2004).

Far-field flow

In order to test the performance of LES, a series of flow rate predictions were made in the merged plume similar to Baines (1983) and Kaye and Linden (2004). The volume flux in a plume is given theoretically by (Kaye and Linden, 2004):

$$Q = \left(\frac{5F}{4\alpha}\right)^{1/3} \left(\frac{6\alpha z}{5}\right)^{5/3} \quad (13)$$

For a value of $\alpha = 0.09$, Kaye and Linden (2004) report that the flow rates above and below the point of coalescence can be written as:

$$z_{below} = 2.28(1 + \psi^{1/3})^{-1/3} Q^{3/5} F_1^{-1/5} \quad (14)$$

and

$$z_{above} = 3.013(1 + \psi)^{-1/5} Q^{3/5} F_1^{-1/5} + z_v \quad (15)$$

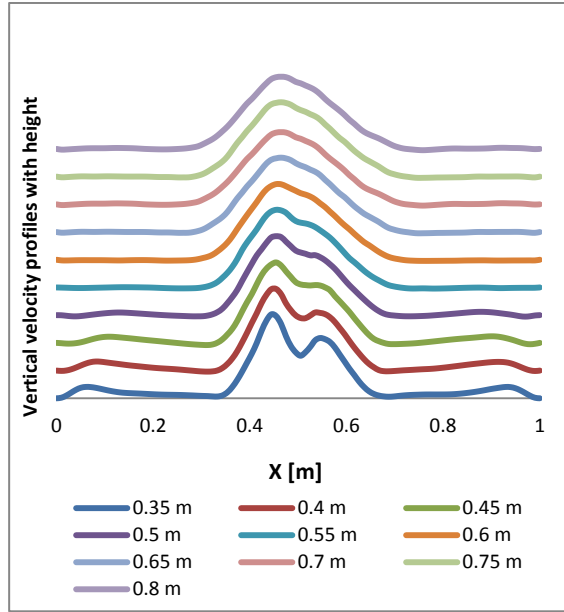


Figure 9: Merging Gaussian profiles of the two plumes

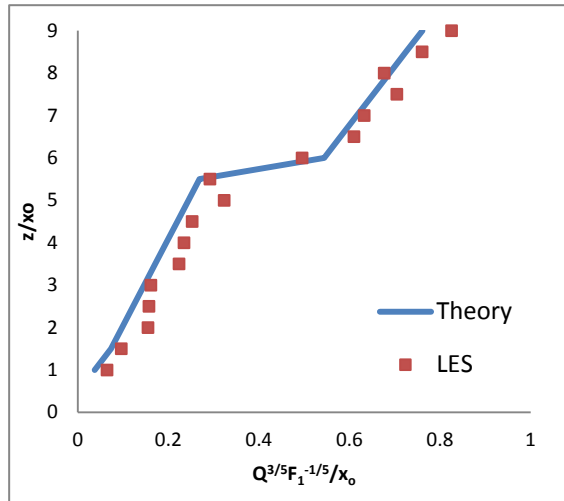


Figure 10: Variation of plume flow rates with height above the heat source (time averaged)

The LES values were calculated using the top-hat velocity profiles in the plumes using the relationships provided by Cook (1998). The LES values clearly provide a good fit to the theoretical predictions (Figure 10). However, it can be seen that the LES simulations tend to over-predict the flow rates. This may be due to over-prediction of the plume entrainment.

CONCLUSIONS

LES has been used to investigate buoyancy-driven natural ventilation resulting from two coalescing plumes. The time averaged interface height that separates the upper warm layer from the lower ambient air is an important feature of such flows. Theory suggests that the interface height should be at a height of 0.677m whereas the LES results suggest a range (i.e. 0.5 to 0.8m) encompassing the theoretical value. This was predicted taking into account the virtual origin phenomena. The LES results of a range, rather than a single value for the interface height, suggests that the interface height remains unsteady throughout the flow process rather than being fixed as theory suggests.

The instantaneous plots of temperature and velocity illustrate the turbulent nature of the plumes. The temperature plots were also helpful in identifying the presence of small coherent structures in the upper warm layer that are thought to contribute to the unsteadiness of the interface plane. Vertical velocity plots in the plumes at different heights helped determining the merging height of the two plumes.

The LES solution of the plume volume flow rate, Q , in the merged plume agreed favourably with the theoretical predictions. The slight over-predictions at some heights is thought to be caused by small variations in the plume entrainment.

It is concluded that LES provides a greater insight than RANS into the turbulent processes that underpin buoyancy-driven natural ventilation and that the results obtained agree well with theoretical models of the same flow.

It is possible that LES could provide valuable insight into the transient evolution and pseudo steady-state behaviour of such flows, which is important in predicting the prevailing air flow directions at inlets and outlets, ventilation rates and temperature distribution in naturally ventilated spaces. The authors are building on the validation work reported here to look at the potential of using LES for larger, more realistic geometries.

REFERENCES

- Abdalla, I. E., Cook, M. J., Rees, S. J. and Yang, Z. (2007). "Large-eddy simulation of buoyancy-driven natural ventilation in an enclosure with a point heat source." *International Journal of Computational Fluid Dynamics* **21**(5-6): 231-245.
- ANSYS_CFX_version_12.1 (2011). www.ansys.com (accessed 08 November 2011). Baines, W. D. (1983). "A technique for the measurement of volume flux in a plume." *Journal of Fluid Mechanics*(132): 247-256.
- Bastiaans, R. J. M., Rindt, C. C. M., Nieuwstadt, F. T. M. and van Steenhoven, A. A. (2000). "Direct and large-eddy simulation of the transition of

two- and three-dimensional plane plumes in a confined enclosure." *International Journal of Heat and Mass Transfer* **43**(13): 2375-2393.

- Cook, M. J. (1998). "An Evaluation of Computational Fluid Dynamics for Modelling Buoyancy-Driven Displacement Ventilation." PhD thesis, (De Montfort University, Leicester.).
- Durrani, F., Cook, M. J. and McGuirk, J. J. (2011). CFD modelling of plume interaction in natural ventilation. *Building Simulation 2011*. Sydney.
- Gladstone, C. and Woods, A. W. (2001). "On buoyancy-driven natural ventilation of a room with a heated floor." *Journal of Fluid Mechanics* **441**: 293-314.
- Hunt, G. R. and Linden, P. F. (2001). "Steady-state flows in an enclosure ventilated by buoyancy forces assisted by wind." *Journal of Fluid Mechanics*(426): 355-386.
- Kaye, N. B. and Linden, P. F. (2004). "Coalescing axisymmetric turbulent plumes." *Journal of Fluid Mechanics* **502**: 41-63.
- Lilly, D. K. (1967). "The representation of small-scale turbulence in numerical simulation experiments." *Proc. of the IBM scientific Computer Symposium on Environmental Sciences. (IBM Form No. 320-1951)*: 195-210.
- Linden, P. F. and Kaye, N. B. (2006). "Interacting turbulent plumes in a naturally ventilated enclosure." *International journal of ventilation* **4**(4): 301-310.
- Linden, P. F., Lane-Serff, G. F. and Smeed, D. A. (1990). "Emptying filling boxes: the fluid mechanics of natural ventilation." *Journal of Fluid Mechanics* **212**: 309-335.
- Nam, S. and Bill Jr, R. G. (1993). "Numerical simulation of thermal plumes." *Fire Safety Journal* **21**(3): 231-256.
- Zhou, X., Luo, K. H. and Williams, J. J. R. (2001). "Large-eddy simulation of a turbulent forced plume." *European Journal of Mechanics - B/Fluids* **20**(2): 233-254.

NOMENCLATURE

a_b	area of the lower opening (m ²)
a_t	area of the upper opening (m ²)
b_G	Gaussian plume width (m)
b_T	top-hat plume width (m)
C_D	coefficient of discharge (-)
C_e	coefficient of expansion (-)
C_p	specific heat capacity (J/kg K)
CFL	Courant number (-)
F	Buoyancy flux (kg/m ² s ²)

g	acceleration due to gravity (m/s^2)
H	total height of computational domain (m)
h	mean interface height (m)
f	loss coefficient (-)
Q	volume flux ($\text{m}^3/\text{s}^1\text{m}^2$)
q	heat source strength (W)
v_G	Gaussian velocity (m/s)
v_T	top-hat value for velocity (m/s)
U_n	normal velocity (m/s)
x_0	source separation (m)
z_m	plume merge height (m)
x, y, z	Cartesian coordinates (m)
α	plume entrainment coefficient (-)
β	coefficient of thermal expansion (-)
ρ	density (kg/m^3)
ε	perturbation in interface height (-)
ξ	normalised interface height (h/H) (-)
ξ_v	location of virtual origin (m)
V	linear velocity (m/s)
ψ	buoyancy flux ratio (-)
ΔP_{loss}	pressure loss across the opening (Pa)
Δt	time step size (s)
Δx	filter width (m)