Numerical studies on natural ventilation flow in an enclosure with buoyancy & wind effects

Yingchun Ji

Institute of Energy and Sustainable Development, De Montfort University, Queens Building, The Gateway, Leicester, LE1 9BH, UK

Abstract

Computational Fluid Dynamics (CFD) has been used to predict natural ventilation flows in an enclosure with openings connected to the environment. The natural driving forces are wind combined with buoyancy. CFD simulations were carried out by varying the strengths of buoyancy under a constant assisting wind effect and the simulation results were compared with the corresponding experimental measurements and analytical studies. The close qualitative and quantitative agreement gave confidence in CFD technique for modelling natural ventilation flows in simple geometry buildings.

Introduction

Natural ventilation has proved to be an effective means to achieve good indoor air quality for occupants with relatively low energy usage in comparison with mechanical ventilation systems. In recent decades, CFD has been increasingly employed in the area of building environmental and natural ventilation strategies and CFD techniques provide much more detailed information on the airflow of interest. With the recent advances in computing power, the process of creating a CFD model and analysing the results is much less labour-intensive, reducing the time and therefore the cost.

Mathematical model

- Volume flux Q through the thermal plume (Fig 1) at different vertical height:

$$x = C^{-3/5} Q^{3/5} B^{-1/5}$$

where B is the buoyancy flux of the thermal plume and C is a parameter dependent upon an entrainment coefficient α, can be expressed by:

$$C = 4^{1/3}\left(1.2\alpha(0.9\alpha)^{1/3}\pi^{2/3}\right)$$

- The relative strength of buoyancy and wind is measured by Fr:

$$Fr = \sqrt{(\Delta/\rho_o)}/(B/H)^{2/3}$$

where Δ is the pressure drop.

- The reduced gravity is defined by:

$$g' = g\Delta\rho/\rho_o$$

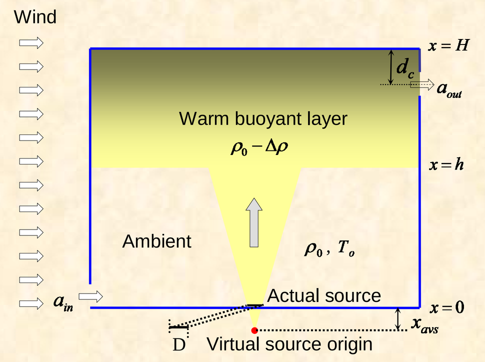

Fig 1 Schematic figure showing the locations of the actual source with diameter D and the virtual source origin.

- Two main assumptions

1. A thermal plume with non-ideal source was proved to be equivalent to a pure point source plume, located at an asymptotic virtual source with a distance $x = -x_{vo}$ below the actual source at $x = 0$ (Fig 1).
2. For cases of purely buoyancy driven flow in [1] and buoyancy combined with wind cases in [2], the volume flow rate through the thermal plume at the interface height is equal to the volume flow rate through the space.

CFD Modelling

- **Commercial CFD code used:** CFX 4.4, structured mesh, Finite Volume Method
- **Turbulence model:** RNG k-epsilon model
- **Boundary conditions:**

  Wall boundary (ceiling, floor, side walls, and the heat source);
  Pressure boundary (inlets/outlets);
  Porous medium boundary (Fig 2).

- **Numerical methods:**

  Pressure coupling is treated by SIMPLEC algorithms and Rhie-Chow method is used to overcome the problem of decoupling due to co-located meshes.

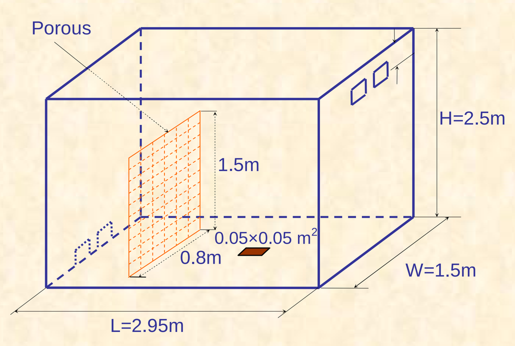

Fig 2 Geometry used for CFD simulations.

Results – qualitative

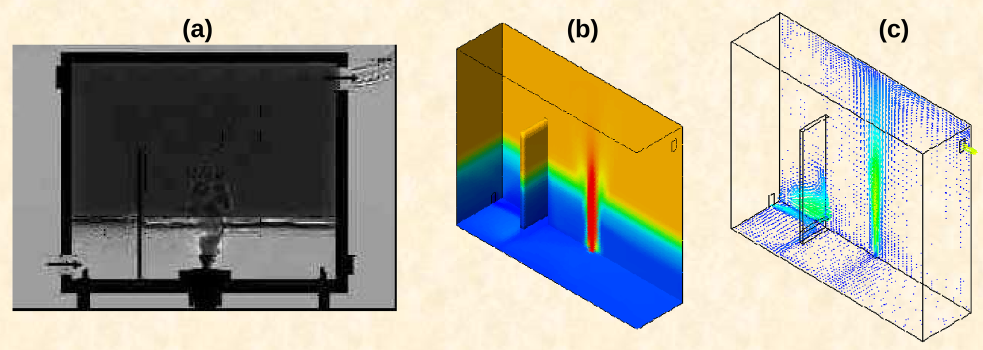

Fig 3 (a) Image from salt bath experiments conducted in [2] showing the density contrast;
(b) shaded contour of the fluid domain showing the temperature contrast, CFD prediction
(c) Speed vectors of the domain showing the uprising thermal plume, inflow and outflow directions.

Results – quantitative

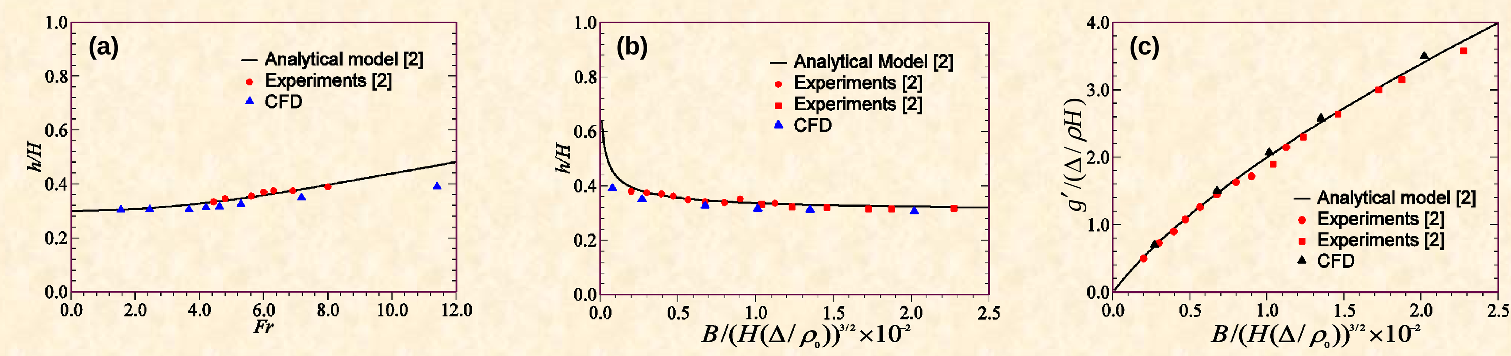

Fig 4 Predictions of analytical model, salt bath experiments ([2], with α=0.083) and CFD for varying heat source strengths, (a) Fr vs h/H; (b) $B/(H(\Delta/\rho_o))^{3/2}$ vs h/H and (c) $B/(H(\Delta/\rho_o))^{3/2}$ vs $g'/(\Delta/\rho H)$.

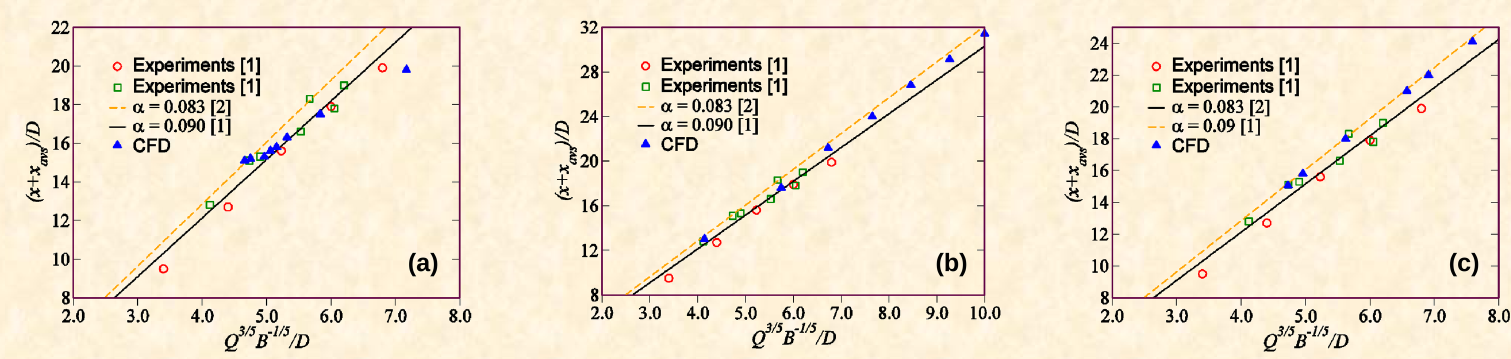

Fig 5 Comparisons of non-dimensional volume flow rates through the space for varying (a) heat source strengths, (b) the relative strength of wind and (c) the ventilation opening size of the flow system.

Conclusions

This work illustrates the ability of CFD for modelling natural ventilation driven by combined wind and buoyancy. The wind force was arranged to assist the buoyancy, and the buoyancy force was varied for different heat loads in the space. The comparisons of volume flow rates through the space, flow patterns, interface positions and the reduced gravities between the analytical model, salt bath experiments and CFD have been made and close agreement was achieved.

References and acknowledgement

[1] Hunt GR & Kaye NG. "Virtual origin correction for lazy turbulent plumes". J.F.M., 435, pp. 377-396, 2001.
[2] Hunt GR & Linden PF. "Steady-state flows in a enclosure ventilated by buoyancy forces assisted by wind". J.F.M. vol. 426, pp. 355-386, 2001.

The work reported here is part of an EPSRC supported project (GR/N37346). The support of the EPSRC is gratefully acknowledged; Dr. Malcolm Cook, Prof. Vic Hanby (De Montfort University) and Dr Gary Hunt (Imperial College of London) are also gratefully acknowledged for their useful comments.

Yingchun Ji is a research student, from China, currently in the third year of his PhD study in building physics at De Montfort University.

Contact:
Tel: 0116 207 8710
yingchun@dmu.ac.uk
www.iesd.dmu.ac.uk