THERMAL COMFORT IN NATURALLY VENTILATED CLASSROOMS: APPLICATION OF COUPLED SIMULATION MODELS

Malcolm Cook[1], Tong Yang[1], and Paul Cropper[2]
[1]Building Energy Research Group, Loughborough University, Loughborough, UK
[2]Institute of Energy and Sustainable Development, De Montfort University, Leicester, UK

ABSTRACT

This paper presents a novel simulation system which couples a computational fluid dynamics program with a model of human thermal comfort and thermo-regulation. The coupled system is used to predict the performance of a buoyancy-driven natural ventilation strategy for a typical school classroom in the UK. Results of the coupled system are compared with uncoupled (CFD alone) approaches to simulation. These comparisons highlight areas where a coupled modelling approach is likely to be beneficial.

INTRODUCTION

Natural ventilation is now frequently used in modern buildings either in isolation or as part of a mixed-mode ventilation strategy. Examples cover a wide range of sizes and applications and include performing arts spaces such as The Lichfield Garrick (Cook and Short, 2005), healthcare facilities such as Braunstone Health and Social Care Centre (Cook and Rees, 2006) and schools (e.g. Firth and Cook, 2010). Natural ventilation not only offers energy savings but can potentially lead to greater occupant satisfaction resulting from higher indoor air quality and more user control through the use of manually openable windows.

Predicting the likely performance of buildings using computer simulation, both in terms of energy use and comfort is now widespread practice in design consultancies around the world. This is particularly true when developing innovative strategies such as those which incorporate natural ventilation. As well as building regulations compliance, computer simulation is being used increasingly to predict thermal comfort and indoor air quality resulting from the temperature and fresh air distribution inside the occupied space. This paper describes work recently carried out to couple a mathematical model of human thermal comfort and thermo-regulation with a commercial computational fluid dynamics (CFD) program and how this has been used to model buoyancy-driven natural ventilation in a school class room.

METHODOLOGY

Thermal Comfort Model

This work uses the IESD-Fiala model (Fiala et al 1999). This is a computer model of human physiology and human thermal comfort which consists of two interactive systems, the passive system and the active system. The passive system represents the physical body and calculates the heat transfer through the various tissue layers of the body and clothing, as a result of the metabolic processes of the body and the local thermal environment surrounding the body. The active system models the body's defence mechanism which seeks to maintain the body's core temperature by regulating blood flow and controlling bodily functions such as sweating or shivering. The outputs from these interlinked system models are predictions of surface temperatures and moisture production rates for 59 regions of the body and resulting predictions of human thermal comfort factors.

The CFD Model

A wide range of commercial CFD codes are currently available, most of which provide facilities for the user to modify and extend the functions of the CFD solver. The CFD code used in this work is ANSYS CFX, version 12 (2011), which was chosen because of the extensive range of customisation options it provides and because it is familiar to the authors. CFX provides various customisation options which include; CFX Command Language (CCL), CFX Expression Language (CEL) and Junction Box routines. In this work CCL is used to pass extra control parameters to the solver, CEL functions are used to dynamically set boundary conditions at the body surface and Junction Box routines are used to extract data from the solver and to exchange data with the IESD-Fiala model during the solver solution cycle. CEL functions and Junction Box routines, written in FORTRAN, are used because they have full access to internal solver variables, subroutines and functions. Junction Box routines are linked to specific events in the CFD solution cycle such as the start of the run or the start of linear solution process. Further details describing how these customisation options are applied are given in Cropper et al. (2010).

The CFX software employs a coupled, fully implicit solver using a transient evolution of the flow from the initial conditions. The physical timesteps used in the transient evolution provide a means of controlling the solution procedure. CFX uses a multi-element type mesh comprising hexahedrals, tetrahedrals, wedges and pyramids. The conservation equations are solved using the Finite Volume method (Versteeg and Malalasekera 2007). Flow variables (velocity, pressure, enthalpy, etc) are defined at the corners of each element which are located at the centre of each control volume used for solving the conservation equations. Solver convergence is deemed to have been achieved when the normalised residual values at the end of an outer iteration fall below a level specified by the user, usually in the range 1.0e-05 to 1.0e-04.

CFX was used in steady-state form in all the simulations reported here. Convergence was controlled using false time-steps of 0.1 seconds on all equations. This method of control sets under-relaxation factors as a function of typical length scales and temperature variations for the flow, thus making this type of control very specific to the flow characteristics being resolved.

In all simulations, air is modelled as a mixture of dry air and water vapour, each represented by the ideal gas equation. Water vapour was included to facilitate the source of moisture emitted from the body surface as predicted by the IESD-Fiala model. As a consequence, buoyancy is represented by the density difference, $\rho - \rho ref$ in the momentum equation, where ρ is the density determined by the ideal gas equation and ρref is a user-defined reference value. In this work, the reference values were taken to be the density of the ambient (incoming) air.

Turbulence is modelled using a k-ω based Shear Stress Transport (SST) turbulence model proposed by Menter (1994) which was chosen for its accuracy in resolving flow field variables within the boundary layer around the human body. Menter (1994) blends a k-ω model near the wall with a k-ε model in the outer region. Buoyancy terms in all of the turbulence transport equations were included. This was done to model the production and dissipation rates of turbulence, due to buoyancy, as closely as possible. The SST model can be used with y-plus values up to 2 making mesh generation more practical than with more traditional models such as low Reynolds number formulations of the k-ω model where y-plus values less than 0.2 are recommended.

A mesh dependency study was carried out using the volume flow rate through the space as the metric. Based on the results of this study it was decided to use a mesh with about 5 million tetrahedral cells and10 prism layers adjacent to the manikin body and walls (see Figure 1). The computational domain is composed of approximately.

Initial tests and previous studies by Zitzmann et al. (2007) showed that modelling radiation between a detailed human model and its surroundings in a simple room using the Monte Carlo method with 2 million tracking histories gave unrealistic temperature distributions on the body surfaces. To avoid this, the Discrete Transfer surface-to-surface radiation model was used with 8 rays. This also led to a saving of 98% in computing time. Emissivity for all room surfaces is assumed to be uniform at 0.95.

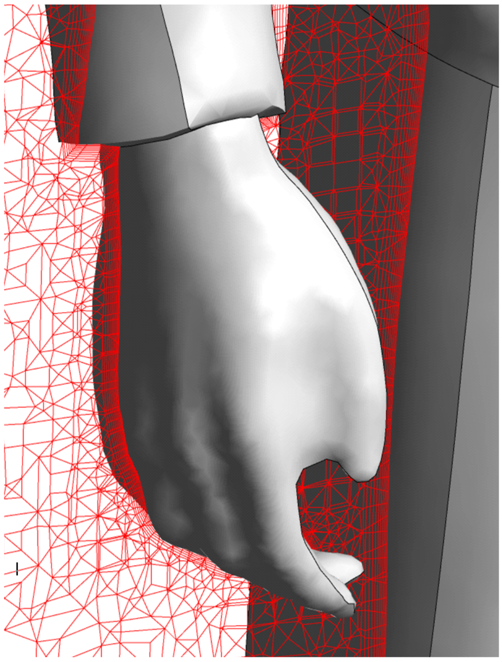


Figure 1. Mesh structure close to body surfaces and simplification of body elements.

The Coupled System

The 59 surface regions of the manikin body surface form the boundary between the body and the CFD domain. Body surface temperatures and sweating rates, predicted by the IESD-Fiala model, are used to set the temperature and moisture production rates of fixed temperature boundary patches. As the CFD solution evolves, the presence of the warm body within the CFD environment affects the local thermal environment surrounding the body. The CFD solver process is interrupted periodically to allow this modified local thermal environment to be applied to the IESD-Fiala model, which in turn provides updated surface temperatures and sweating rates. The temperature and moisture production rates at the CFD boundary patches are updated and the CFD solver process resumed. This exchange of data is repeated until the difference in mean body temperature (average of 59 body surface temperatures) between two consecutive data exchanges falls below a predefined value, i.e. when changes in body surface temperature are minimal.

Further details describing the model coupling and data exchange procedures is presented in Cropper et al. (2010).

When a coupled simulation is first started, the IESD-Fiala model supplies initial surface temperatures and moisture production rates, based on user-defined initial conditions, to set the CFD boundary conditions at the body surface. No data exchange then takes place until the mass and momentum RMS residual values fall below a user-defined threshold, typically 1e-04. This enables the flow in the CFD model to become established following the specification of initial conditions (zero flow in this case). Once this threshold has been achieved data exchanges take place at regular intervals until the RMS residuals fall below a second user-defined threshold, typically 1e-05, or until the difference between the average of the body surface temperatures following two consecutive data exchanges is less than 0.001K. Each time a data exchange takes place, additional errors are induced in the energy balance equations due to changes to the CFD boundary conditions. These errors are seen to quickly reduce over the following few iterations before the downward trend is resumed and before the next data exchange takes place. The magnitude of the errors induced by each data exchange is seen to reduce as the coupled system approaches convergence. Following each data exchange, a user-defined number of CFD outer iterations, typically eight iterations, must be completed before the next data exchange takes place. This enables the CFD solution to recover from instabilities caused by changes in the boundary conditions.

The flow chart in Figure 2 gives an overview of the simulation process and the criteria used for exchanging data. The decision regarding when to stop the simulation process is left to the judgement of the CFD operator and is no different to traditional (uncoupled) use of CFD. On issue of the user command to stop the simulation, the process in the flowchart will terminate upon completion of the next "Perform CFD Iteration" process.

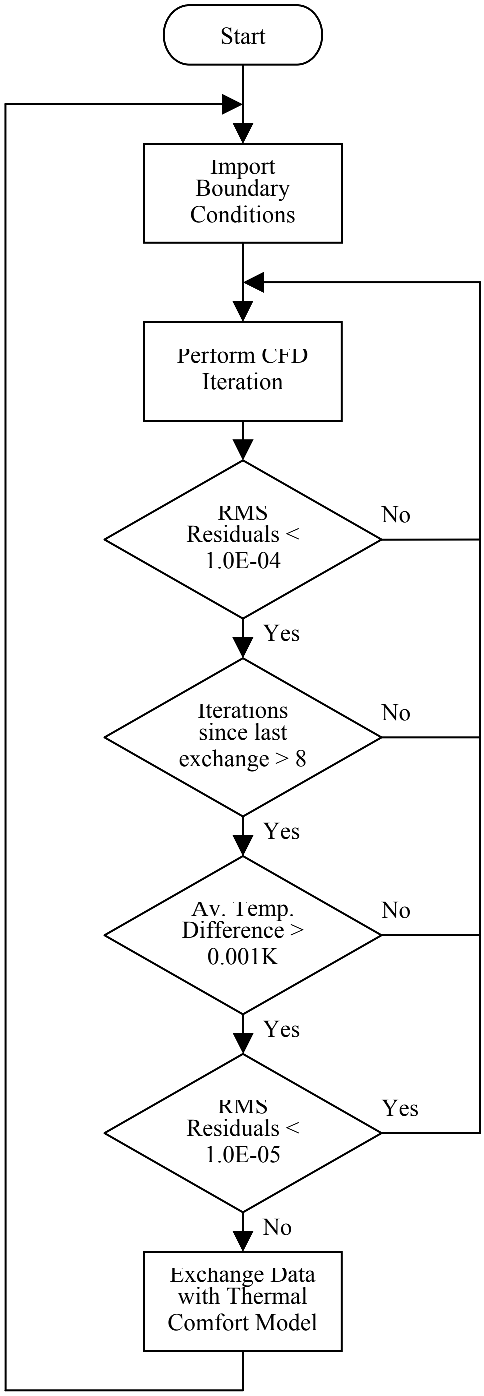


Figure 2. Flowchart showing an overview of coupled simulation system

CASE STUDY

The space under consideration is a secondary school classroom which has been designed for stack-assisted cross ventilation (Figure 3). The classroom measures 8.25m x 6.4m in plan with a floor to ceiling height of 2.7m. Natural ventilation is provided by two opening windows (each of 0.375m^2) and a vertical stack with cross section area equal to 0.64m^2. The stack is 7.8m above the classroom ceiling with openings at the top and bottom of 0.64m^2. The openings were sized to give an air change rate of approximately 10 ac/h. In the experience of the authors, this amount of ventilation is necessary during summer to provide sufficient passive cooling.

Boundary conditions imposed at the window inlets enable air to flow into or out of the domain, depending on the local pressure distribution established by the naturally occurring temperature differences. Mathematically, this so-called 'opening' type boundary condition can be represented using a relative pressure, p_{spec} and a loss coefficient, f, so that the driving pressure at the opening, $p_{opening}$, is given by:

$p_{opening} = p_{spec} + \frac{1}{2} f \rho U_n^2$ (for outflows)

$p_{opening} = p_{spec} - \frac{1}{2} f \rho U_n^2$ (for inflows) where U_n is the magnitude of the velocity component normal to the plane of the opening.

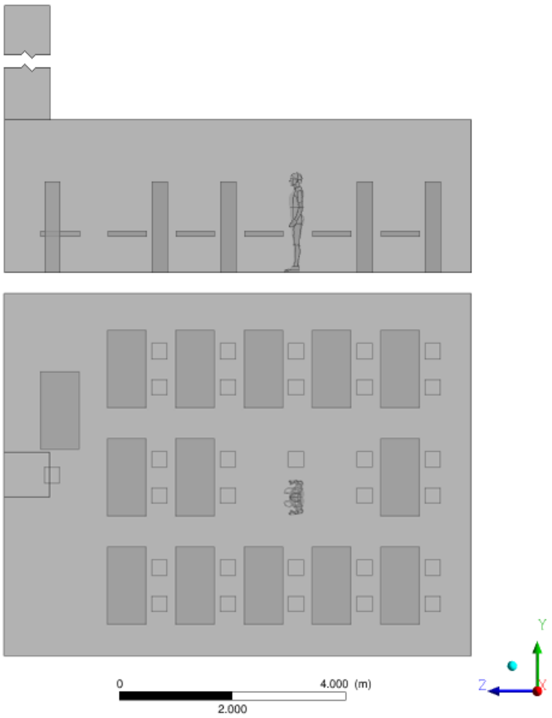


Figure 3 Section and plan of case study geometry.

A loss coefficient of 2.69 was used at all openings. This is resembles flow through a sharp-edged orifice which is commonly used in these situations in the absence of manufacturer's data.

The room comprises 31 occupants and no other heat sources. One of the 30 occupants is represented as a computational manikin (CM) whereby the geometry has been generated using the human geometry generation tool, Poser (2011). For more information, see Yang et al. (2007). The other 30 occupants are modelled using rectangular blocks in an attempt to provide a side-by-side comparison of the methods for representing occupants. The CM is coupled to the IESD-Fiala model.

Two outside air temperatures were considered, 21°C and 28°C. In each case, the relative humidity was set to 50%. For each case, two simulations were conducted: uncoupled (CFD alone), and the fully coupled system.

In all uncoupled CFD cases, boundary conditions imposed on the rectangular block and the CM are 90W. In the coupled system, the CM takes values for temperature and moisture flux from the IESD-Fiala model. The boundary condition imposed on the surrounding surfaces (floor, ceiling and walls) was a U-value of 0.32W/m^{2}K.

The iteration process was monitored using 5 points: one at 0.2m above the CM head, another between the rectangular blocks at a height of 1.5m, one at the centre of each window opening, and one at the centre of the stack outlet.

RESULTS

CFD simulations were considered to be converged when the normalised residuals for all equations fell below 1E-05 and the values of all variables at the monitoring points were constant. All results presented here were considered to have converged according to this criteria.

For each case, results are tabulated (Table 1) to show the MRT (mean radiant temperature), the DRT (dry resultant temperature), thermal sensation, and the PPD (predicted percentage dissatisfied). The thermal sensation is given by the PMV (predicted mean vote) in the CFD simulations and by the dynamic thermal sensation (DTS) in the coupled system. The table also shows the ventilation rate induced by the buoyancy forces.

Table 1
Numerical predictions comparing uncoupled CFD with coupled CFD

	Uncoupled CFD Simulation		**New Coupled System**	
Outside temperature (°C)	21.0	28.0	21.0	28.0
Ventilation rate				
l/s/p	15.0	14.2	14.8	14.8
ac/h	11.0	10.4	10.9	10.8
MRT (°C)	27.63	33.97	28.57	34.48
DRT (°C)	26.18	32.71	26.70	33.13
Thermal sensation (-)	0.23 (PMV[1])	2.15 (PMV[1])	0.41 (DTS[1])	1.97 (DTS[1])
PPD (%)	7.98	82.38	8.56	75.55

1. PMV is predicted by CFD model. DTS is predicted by IESD-Fiala model

Both simulation techniques successfully predicted buoyancy-driven natural ventilation (Figure 4). These results show a warm layer of buoyant air in the upper part of the classroom which drives a flow upwards through the ventilation stack and out of the building. The resulting pressure difference at the windows draws cooler, outside air, into the classroom which forms a layer of fresh air in the lower part of the room. The ventilation flow rates, in all cases, are in the range 10.4 – 11.0 ac/h which agrees with expectations based on the opening sizes and loss coefficients modelled.

The warm air layer responsible for driving this ventilation flow forms due to the warm bodies in the room (see figures 5a and b). The uncoupled CFD modelling solution (Figure 5a) clearly shows similar temperature predictions for both modelling approaches. This is expected as the surface areas of both representations are identical. In contrast, the new coupled modelling approach (Figure 5b) predicts a lower surface temperature for the CM (the surface temperature of the other occupant representations are unchanged). The lower surface temperature, depicted more clearly in figure 6, is predicted by IESD-Fiala model in response to the higher air temperatures and surrounding surface temperatures in the classroom. Under such conditions, the 'active' module within the IESD-Fiala model responds to the warmer conditions by increasing sweat rate and blood flow to the skin surface. Both of these responses reduce the body surface temperature as required.

It is interesting to speculate what may happen to the ventilation rate in the case where multiple instances of the coupled model are included. Lower body temperatures would lead to a reduced buoyancy driving force and *could* result in a reduction in the natural ventilation rate. What may happen, however, is the reduced ventilation rate would lead to a higher temperature in the space which would eventually re-establish a higher flow rate.

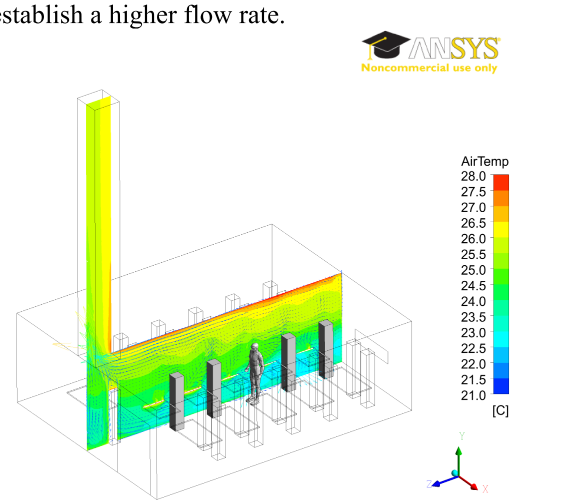
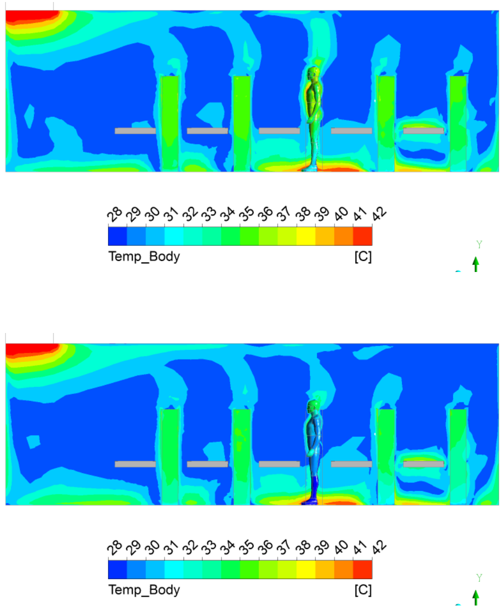


Figure 4 Vertical temperature distribution over a plane passing through the stack. Outside air temperature = 21°C.

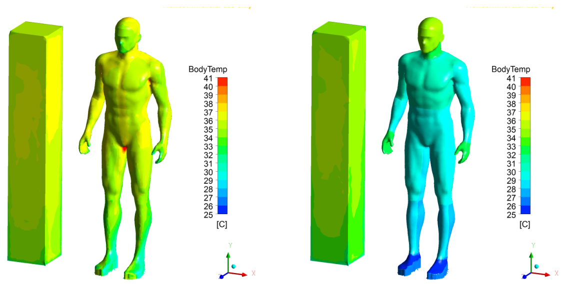
For both the uncoupled and the coupled system, the thermal sensation, measured according to Fanger's 7-point thermal sensation scale, lies between 'neutral' and 'slightly warm' for the 21°C case, and about 2 ('warm') for the 28°C case. The similarities in results for the two modelling approaches gives confidence that the new coupled system is not introducing significant errors into the simulation procedure. The differences in thermal sensation and PPD between the two systems most likely result from the ability of the IESD-Fiala model to thermo-regulate (in this case by sweating and increasing blood flow to the skin), thus leading to the prediction that occupants are likely to be more comfortable.



(a)

(b)

Figure 5 Body surface and air temperature predictions for (a) uncoupled CFD and (b) coupled CFD simulations. Outside air temperature = 21°C.



(a) *(b)*

Figure 6 Body surface temperature predictions for uncoupled CFD (a) and coupled CFD (b) simulations. Outside air temperature = 21°C.

CONCLUSIONS

This study demonstrates the application of a novel simulation system for predicting thermal comfort and human thermoregulation in naturally ventilated spaces. The system comprises an advanced model of human heat transfer and thermal comfort embedded within a commercially available CFD program. The development of user-customisation routines within the CFD software to extend its functionality has enabled specific parameters to be supplied to and extracted from the solver during the solution cycle in a form suitable for exchange with the IESD-Fiala model.

The coupled system has been applied to the case study example of buoyancy-driven natural ventilation in a classroom. This is a particularly appropriate test for the coupled model as the predicted airflow characteristics rely on the accurate prediction of local temperature differences between the human body surface and the surrounding air.

The comparison of the CFD alone (uncoupled) model with the coupled simulation system highlight the benefits of the integrated system, namely, more accurate body surface temperatures, leading to a more accurate buoyancy driving force. The coupled system also provides the advantage of predicting thermal comfort metrics which are based on a more detailed representation of the human body's thermal heat transfer and physiology.

Future work will consider multiple instances of the IESD-Fiala model which will enable the team to investigate in detail the effect of a thermoregulated human body on the buoyancy forces and hence the natural ventilation flow rate. Multiple instances will also facilitate better prediction of thermal comfort effects resulting from person-to-person radiation. The team has also added breathing effects to the computational manikin to enable the assessment of indoor air quality (IAQ) and ventilation effectiveness. In addition to thermal comfort, IAQ issues are of great interest in school classrooms which are intended to be well designed spaces with ventilation strategies capable of providing a healthy, stimulating and productive environment.

ACKNOWLEDGEMENTS

The authors wish to acknowledge the Engineering and Physical Sciences Research Council (EPSRC) for their support for this research through grant ref. EP/C517520/1.

The authors also wish to acknowledge the technical support provided by Dr Yehuda Sinai (formerly of CFX Technical Services) and Mr Chris Staples, CFX Technical Services, ANSYS Europe Ltd.

The authors would like to thank Dr Yi Zhang, De Montfort University, for his work in translating the IESD-Fiala model into the Java language, and Rehan Yousaf (Loughborough University) for his assistance with mesh refinement.

REFERENCES

Abdalla, I. E., Cook, M. J., Rees, S. J. & 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, vol. 21, Issue 5 & 6, 231-245.

Ansys http://www.ansys.com/, accessed May 12th 2011.

Cook MJ and Rees SJ. 2006, Low Energy Design of Community Healthcare Buildings: A Case Study, Proc. World Renewable Energy Conf, Florence.

Cook MJ and Short CA, 2005, Natural Ventilation and Low Energy Cooling of Large, Non-Domestic Buildings - Four Case Studies, The International Journal of Ventilation, Vol 3 No 4, pp 283-294.

Cropper, P.C., Yang, T., Cook, M.J., Fiala, D. and Yousaf, R., 2010 'Coupling a model of human thermoregulation with computational fluid dynamics for predicting human-environment interaction', Journal of Building Performance Simulation, Vol. 3 Number 3, pp.233-243.

Fiala D, Lomas KJ and Stohrer M, 1999, A computer model of human thermoregulation for a wide range of environmental conditions: The passive system, Journal of Applied Physiology, Vol. 87 (5), pp 1957-1972.

Firth SK and Cook MJ, 2010, Natural ventilation in UK schools: design options for passive cooling, Network for Comfort and Energy Use in Buildings, Proceedings of Windsor 2010 conference: Adapting to Change: New Thinking on Comfort, Cumberland Lodge, Windsor, UK, 9-11 April.

F. R. Menter. 1994, Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA Journal. Vol. 32. pp. 1598–1605. 1994.

Poser http://poser.smithmicro.com/, accessed May 11th 2011.

Versteeg HK and Malalasekera W, 2007. An introduction to computational fluid dynamics – the finite volume method, 2nd ed. Pearson Education Ltd., Harlow, Essex, UK

Yang T, Cropper PC, Cook MJ, Yousaf R and Fiala D, 2007, A new simulation system to predict human-environment thermal interactions in naturally ventilated buildings, Proc. 10th International Building Performance Simulation Association Conference and Exhibition (BS2007), Beijing, China, pp751-756.