CFD reliability issues in analysis of naturally ventilated buildings

J.M. Horan
Urban Institute Ireland, University College Dublin, Ireland

D.P. Finn
Department of Mechanical Engineering, University College Dublin, Ireland

ABSTRACT

The potential for error when using computational fluid dynamics (CFD) for investigating internal building airflows continues to be a critical issue in building simulation analysis. This topic is assessed in the current paper by examining the ability of a proprietary CFD code to simulate buoyancy and forced airflow regimes, typical of a naturally ventilated building. This issue is motivated by an ongoing research project, aimed at examining the relationship between external microclimate and internal building comfort, where CFD constitutes a major analytical tool. Two experimental case studies from the literature are employed as benchmarks by which the CFD code is assessed. First, the Cheesewright study, which is based on pure buoyancy cavity flow, is considered. Results from this investigation indicate that structured meshes are not only less dependent on mesh density, but also give consistent convergence and accuracy when coupled with the k-ε and k-ω turbulence models. Second, the Neilson study, which is based on forced airflow data, gave broadly similar findings, however, slightly better prediction of peak air velocities were observed when the k-ω turbulence model was used. The paper concludes by discussing the relevant measures that must be considered when applying CFD methods to a full-scale naturally ventilated building.

1. INTRODUCTION

World energy consumption is expected to double by the middle of the current century if global energy usage continues unabated (IEA, 2004). The increasing popularity of building air-conditioning is in part responsible for a portion of this trend. In the UK alone, annual sales of commercial air-conditioning units increased from 55,000 in 1988 to over 220,000 units in 2001, 45% of which were installed in commercial offices (ACE, 2003). Accordingly, there is increasing concern about the sustainability of this growth.

In order to address these concerns, low energy cooling approaches, such as natural ventilation, are receiving increased interest in the building energy sector. Not only does natural ventilation offer potential energy savings, but surveys have also found a higher level of satisfaction with the internal environment among occupants of naturally ventilated buildings. In one study discussed by Clements-Croome (1997), a survey of six office buildings found symptoms of sick building syndrome (SBS) to be more prevalent in mechanically ventilated buildings than naturally ventilated buildings. Wong and Huang (2004) also report that occupants exhibited more SBS symptoms after sleeping in air-conditioned spaces, compared to naturally ventilated ones.

A key issue in the use of natural ventilation is assessing the quality of indoor air (IAQ). One factor that can adversely affect IAQ is external microclimate fluctuations which have the potential to cause significant variation in internal air flow patterns. A common tool used in assessing internal building air movement is Computational Fluid Dynamics (CFD), which allows precise resolution of internal space variables to be achieved. However, a key issue when using CFD, is the specification of mesh and turbulence models, which play a significant role in the integrity of the overall computational solution.

Baker et al., (1997) note that any CFD model constitutes the culmination of a large number of assumptions and approximations, thereby giving rise to potential accumulated error in any building energy analysis. Iannone (2001) observes that while numerous comparisons between numerical simulations, scale-model experiments and real system measures have shown that CFD results are generally in good agreement with experimental data, uncertainties due to boundary conditions can significantly affect simulation accuracy. Ding et al., (2005) found good comparison between CFD predictions and small-scale experimentation for airflow in a naturally ventilated double skin façade building, however Ding does not address the validity of the application of such models to full-scale building analysis. Furthermore, Luo et al., (2004) notes that little work has addressed validation of CFD codes when used in full-scale analysis of naturally ventilated buildings.

Before addressing the greater objective of this research, aimed at examining the relationship between external microclimate and internal building comfort, where CFD constitutes a major analytical tool, it was deemed critical to consider issues pertaining to the integrity of use of CFD. In this paper, this is examined by rigorous comparison with a range of experimental data, including buoyancy cavity flow and forced flow. In addition, the CFD predictions are compared to large-scale experimental building data recorded over an extended period of time in an occupied functioning naturally ventilated building.2. CHEESEWRIGHT AND NEILSEN CASES

Airflows in naturally ventilated buildings generally result from wind and buoyancy driven flows. Therefore, a first step to investigating the accuracy of CFD for analysing natural ventilation flows, was to assess each flow type individually. This was undertaken by assessing the ability of the CFD code to predict airflows similar to experimental data obtained from two literature sources, Cheesewright et al., (1986) and Nielsen (1990).

2.1 Cheesewright Model

In this experiment, the authors describe a two-dimensional buoyancy induced airflow that occurs in an enclosed cavity. The induced airflow is a result of a temperature difference between opposite walls of the cavity. The cavity has a height 2.0m and a width of 0.5m as shown in Figure 1. The hot and cold walls were constructed from single sheets of aluminum alloy, with insulated front and back walls. Laser Doppler anemometry was used to measure internal air velocities. A series of heater elements were attached to the back of the hot plate such that the power supplied to individual heaters could be adjusted to maintain a constant uniform plate surface temperature. The cooling effect was supplied to the cold wall by forcing air through a series of channels on the back of the cold wall plate. The airflow through individual channels was controlled to ensure a constant uniform plate surface temperature.

The Cheesewright data is divided into three sets, each corresponding to a different temperature difference between the hot and cold walls. The data used in this work is for a temperature difference of 45.8 K between the hot and cold walls and consists of 2-D velocity and temperature measurements.

There were two main issues to be considered when creating a CFD model for comparison with the Cheesewright data. First concerns the choice of mesh which can be either structured or unstructured, and second is the issue of turbulence model. Various permutations of mesh type and turbulence model were investigated to assess their influence. CFX 4 was used to develop 2-D structured hexahedral meshes while CFX 5

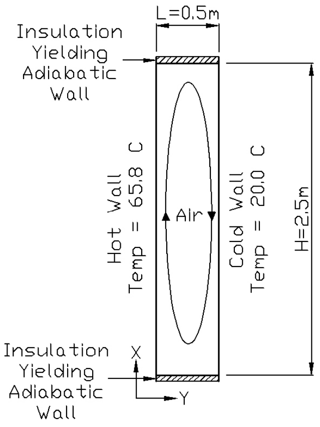

Figure 1: The air cavity used in the Cheesewright experiments (Cheesewright et al., 1986).

was used to develop 2-D unstructured tetrahedral meshes. Mesh geometries were then imported into CFX 5 where the models were built and solved.

2.2 The Nielsen Model

The Nielsen experiments were carried out as part of an IEA Annex 20 project, aimed at developing test case data for the validation of CFD codes in building airflow analysis (Nielsen, 1990). Figure 2 shows the geometry of the test case model which is representative of a typical ventilated space. The test data are subdivided into an isothermal air distribution test case and a non-isothermal test case. The isothermal test case was utilised in the current study.

For the current study, data from a model with the following parameters was used: height, H, and width, W, each 3.0 m, a length L of 9.0 m. The air inlet height, t, is 0.48 m while the air outlet height, h, is 0.168 m. The inlet velocity of the air at 20^{o} C was 0.455 ms^{-1} with a turbulence energy dissipation rate, ε, of $6.59x10^{4}$ $m^{2}s^{-3}$ and turbulent kinetic energy, k, of $4.97x10^{4}$ $m^{2}s^{-2}$.

The experimental measurements obtained are presented in several detailed graphs. Velocity measurements are given along two vertical lines at x = H and x = 2H, and along two horizontal lines, y = h/2 and y = H – h/2.

As with the Cheesewright model, there were two main parameters to be investigated in the Nielsen model, the mesh type employed and the turbulence model used. Again CFX 4 was used to construct 2-D structured meshes while CFX 5 was used to constructed 2-D unstructured meshes. Various different mesh densities were investigated. The meshes were imported into CFX 5 where the models were developed and solved.

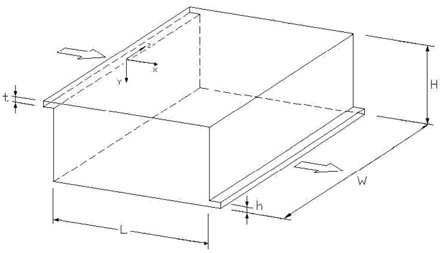

Figure 2: The Nielsen Test Case Space (Nielsen, 1990).

3. FULL-SCALE BUILDING

In order to obtain data for actual airflows experienced in naturally ventilated buildings, a building-monitoring program has been undertaken. The data obtain from this monitoring program will be used for direct comparison with the CFD model predictions.

The building chosen for the monitoring program is the Urban Institute located at University College Dublin (see Fig. 3). The building, which is located in a suburban city environment, is primarily constructed using mass concrete with exposed floors, walls and ceilings. The interior of the building is arranged into a series of interconnecting corridor office spaces distributed around a reception area (lower level) and a central atrium space (upper level). The ground floor consists of a reception area, a lecture room, an atrium space, a series of offices, rest rooms, a laboratory and an open staircase. The first floor contains an open staircase, a series of offices, rest rooms and an overlooking balcony into the atrium space. The reception and atrium

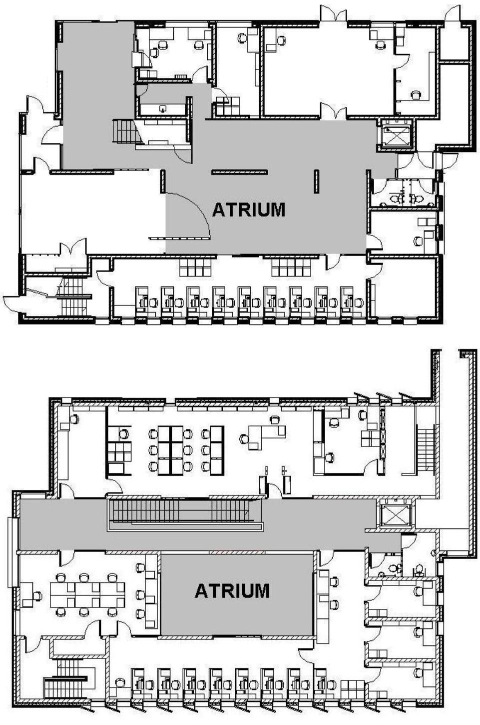

Figure 3: Ground floor (top) and first floor plans (bottom) of the UI building with areas of interest shaded.

spaces primarily serve as informal meeting and discourse areas. In this study, the combined reception and central atrium spaces over both levels are considered.

A full weather station, installed on the roof of the building provides data on external localised climate conditions. Data is also obtained from a weather station located at Dublin airport (15 km distant) operated by the Irish metrological service. Various sensors are positioned throughout the interior of the building to obtain measurements of air temperature, wall surface temperature and humidity. Thermo-anemometers, used to measure low-level air velocities as well as air temperature, are mounted on a specially designed 7m high movable tower. The thermo-anemometers can be moved in a matrix grid pattern within the atrium space to obtain localised temperature and velocity data.

The main sources of ventilation for the atrium space are the main entrance door, small ground level windows, other internal spaces and two natural ventilation vents connected to the outside on the lower level, and five roof top automated windows on the upper level. Continuous monitoring of the interior door openings is challenging to track without event counters, however as the doors are spring mounted they default to their closed position, whilst all ground floor windows remain closed. Building ventilation is effected by a combination of the two permanently open side vents and the roof-top windows. Automatic ceiling fans are mounted beside the automated roof-top windows. Window opening, vent opening, heating system operation and fan operation are all tracked via a building monitoring system while vent airflow is monitored using a thermo-anemometer. It is intend to use data obtained from the building while the fans are off to ensure that any airflow in the space is a result of natural forces only.

4. CFD RESULTS

4.1 Cheesewright Model

In order to compare the results obtained using structured and unstructured meshes several models were considered. Models were meshed using a similar number of structured or unstructured elements and then solved using the three turbulence models, k-ε, RNG k-ε and k-ω. Several different mesh densities were investigated. It was found that although the unstructured meshes sometimes gave better results than the structured case, these results were highly dependent on the mesh density. The structured meshes however exhibited a weaker dependency on mesh densities and were found to perform better. All structured meshes solved relatively quickly with RMS residuals falling linearly to $1x10^{-6}$. Many of the unstructured meshes failed to converge exhibiting residuals generally not less than $1x10^{2}$ sometimes with oscillatory tendencies. It was concluded that structured meshes provided the best modelling capabilities for solely buoyancy driven airflows.

Figure 4 shows predicted CFD velocities, for a structured and unstructured mesh, plotted against measured data from the Cheesewright experiments. Figure 5 shows the error associated for a structured mesh for each of the three turbulence models.

Results obtained from identical structured mesh models, solved using each of the three turbulence models, indicated that the k-ε and k-

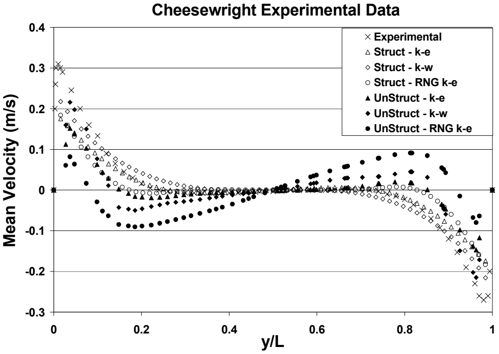

Figure 4: Mean velocity at mid-cavity vertical height.

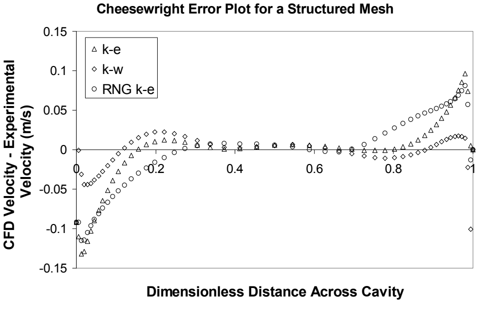

Figure 5: Error specification of different turbulence models for Cheesewright data at mid-cavity vertical height.

ω models performed well, while the RNG k-ε performed poorly. The k-ε was the most consistently reliable but the k-ω at times provided the best results. Convergence of results was similar for both turbulence models.

4.2 Nielsen Model

From the Cheesewright study, it was evident that the structured models gave better velocity profiles than the unstructured models, irrespective of turbulence model used.

For the Nielsen case, both structured and unstructured meshes were investigated. It was found that the structured meshes performed best. Structured meshing was chosen for the mesh type and the choice of turbulence model was then investigated. Three turbulence models were considered k-ε, RNG k-ε and k-ω. It was found that both k-ε and k-ω performed well, however the RNG k-ε gave poorer results. Comparison of the k-ε and the k-ω models reveals little difference. Figure 6 shows dimensionless velocity in the x-direction at x/H= 2.0 (where U_o is grille average inlet velocity) for one set of structured mesh simulations. It is evident that all three turbulence models perform well with k-ε and k-ω performing marginally better than RNG k-ε. Figure 7 calculates the error limits for the three turbulence models, where the performance of the k-e model can be easily seen.

4.3 Discussion: Cheesewright and Nielsen

Based on the analysis of the Cheesewright and Nielsen model results, it was decided to proceed onto the full building simulations using a structured mesh with the k-ε turbulence model. The selection of the structured mesh was straight forward as it performed best in almost all of the simulations carried out for both the Cheesewright and Nielsen experiments. The selection of the turbulence model however was more difficult. It was felt that both the k-ε and k-ω performed well for the Cheesewright and Nielsen experiments. The k-ε proving the more reliable as the k-ω was found to be more mesh dependent than the k-ε. However the k-ω proved to be the better of the two models at predicting peak velocities in the Cheesewright model. It has been decided to begin the modelling of the full building using a structured k-ε model, based on the earlier discussion and also based on similar findings by other authors for natural ventilation modelling work (Gan and Riffat, 2004; Linden, 1999). Once the mesh and model have been refined to satisfactory degree, the k-ω model shall also be used to solve the model and allow comparisons between the k-ε results, the k-ω results and the experimentally measured values.

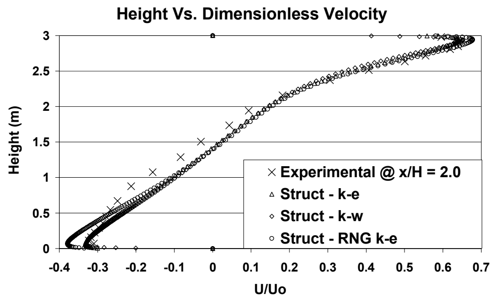

Figure 6: Dimensionless velocity (U/U_0), along central vertical axis at x/H = 2.0 m.

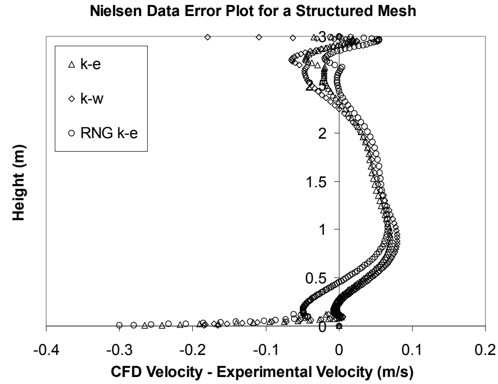

Figure 7: Error specification of different turbulence models for Nielsen data along central vertical axis at x/H = 2.0 m.

4.4 Full-Scale Building Modelling

Development of a 3-D CFD model of the naturally ventilated building under investigation is currently underway. An initial structured mesh model has been built in CFX 4 and been imported into CFX 5 where the model is under development. Data from the building monitoring scheme is being used for initial conditions and boundary conditions for the CFD model. CFD results obtained from the models shall be compared to measurements taken as part of the monitoring program to validate the model.

5. CONCLUSIONS

Experimental data from two experiments, documented in the research, was used to investigate the capabilities of a commercial CFD code. One experiment contained flow measurements for purely buoyant airflow while the other contained data for forced airflow. CFD models were developed to simulate the experiments and to allow comparison between predicted and experimental measurement. It was found that a structured mesh outperformed an unstructured mesh in terms of both solution convergence and predicted airflows.

Of the three commonly used turbulence models investigated, k-ε, k-ω and RNG k-ε, it was found that both k-ε and k-ω performed well with the RNG k-ε performing poorly for the airflows investigated.

Development of a 3-D CFD model of the airflow inside an operational naturally ventilated building is currently under way. The model will have a structured mesh and will use the k-ε turbulence model. Experimental data from the building-monitoring program will be used as boundary conditions for the model and for comparison with the CFD predictions.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support of the Urban Institute, Ireland and the President’s Award, University College Dublin.

INTERNET SOURCES

ACE, Association for the Conservation of Energy (2003). Energy efficiency in offices: assessing the situation. http://www.ukace.org/pubs/reportfo/offices_1.pdf.

IEA, International Energy Agency (2002). International Energy Outlook 2002. International Energy Agency. Available at: www.eia.doe.gov/oiaf/ieo/index.html.

REFERENCES

Baker, A.J., R.M. Kelso, E.B. Gordon, S. Roy and E.G. Schaub, 1997. Computational Fluid Dynamics: A Two Edged Sword, ASHRAE Journal, August, Vol, No, pp.51-58.

Cheesewright, R., K. King and S. Ziai, 1986. Experimental Data for the Validation of Computer Codes for the Prediction of Two-Dimensional Cavity Flows. Significant Questions in Buoyancy Affected Enclosure or Cavity Flows - Winter Annual Meeting of the ASME, Dec 7-12, pp. 75-81.

Clements-Croome, D., 1997. Naturally Ventilated Buildings – Buildings for the Senses, the Economy and Society, E & FN Spon.

Ding, W., Y. Hasemi and T. Yamada, 2005. Natural ventilation performance of a double-skin façade with a solar chimney. J. Energy & Buildgs. 37, pp. 411-418.

Gan, G. and S.B. Riffat, 2004. CFD modelling of air flow and thermal performance of an atrium integrated with photovoltaics. J. Buildg & Environt. 39, pp.735-748.

Iannone, F., 2001. Natural Ventilation and Sustainability Designing with Computational Fluid Dynamics. Clima 2000, Napoli, September.

IEA, International Energy Agency, 2004. World Energy Outlook 2004. International Energy Agency.

Linden, P.F., 1999. The fluid mechanics of natural ventilation, Annual Reviews of Fluid Mechanics 31 pp. 231–238.

Luo, S., J. Heikkinen and B. Roux, 2004. Simulation of air flow in the IEA Annex 20 test room—validation of a simplified model for the nozzle diffuser in isothermal test cases. J. Buildg & Environt. 39, pp. 1403-1415.

Nielsen, P.V., 1990. Specification of Two-Dimensional Test Case. International Energy Agency (IEA) Annex 20 Internal Report. Aalborg University. ISBN ?

Wong, N.H. and B. Huang, 2004. Comparative study of indoor air quality of naturally ventilated and air-conditioned bedrooms of residential buildings in Singapore. J. Building and Environment. 39, pp. 1115-1123.