Numerical Simulation of Convective Airflow in an Empty Room

Kana Horikiri, Yufeng Yao and Jun Yao

Abstract—Numerical simulation of airflow inside an empty room has been carried out for a forced convection, a natural convection and a mixed convection respectively, by using a computational fluid dynamics approach of solving the Reynolds-averaged Navier-Stokes fluid equations. Two-dimensional model was studied at first; focusing on the grid refinement, the mesh topology effect, and turbulence model influences. It was found that structured mesh results are in better agreement with available experimental measurements for all three scenarios. Further study using a three-dimensional model has shown very good agreements with test data at measuring points. Furthermore, present studies have revealed low-frequency flow unsteadiness by monitoring the time history of flow variables at measuring positions. This phenomenon has not yet reported and discussed in previous studies.

***Keywords*—**Computational fluid dynamics, forced convection, mixed convection, natural convection

I. Introduction

Indoor environment design requires details of air velocity/temperature distributions, relative humidity maps, contaminant concentrations, and turbulence levels. Most indoor airflows are actually rather complicated, and often driven by both pressure gradient and thermal buoyancy forces. There are three typical convection modes; i.e. forced convection like spring free cooling flow near the ceiling, natural convection as winter heating by radiators, and mixed convection as summer cooling using an air-conditioning unit. Despite the challenges in predicting the airflow precisely, both experimental measurements and computational simulations have been used in the past. Most experiments adopt a full-scale test chamber to setup an artificial environment to isolate the space from the external. While this permits the controllable flow and thermal boundary conditions, the cost would be high and turn-around testing period is normally very long.

Earlier studies applied analytical models that are derived from fundamental governing equations of heat and fluid flow, mainly mass, momentum energy and chemical species. Most analytical models use simplified geometry and thermodynamic boundary conditions in order to carry out ‘quick’ solution. For example, Fitzgerald and Woods studied the influences of stacks on airflow patterns and stratifications driven by natural convection in a 2D model room with two openings [1]. Another method was proposed by Mazumdar and Chen applying superposition concept and variable separation method [2]. These methods have been applied for analyzing various problems, including thermal buffering of naturally ventilated building [3], time lag prediction and indoor contaminant fluctuation reduction [4], among many others.

With the advancement of computer power and numerical method, numerical approach using a computational fluid dynamics (CFD) has been widely used to model the indoor airflows and other relevant applications [5-8]. There are significant progresses being made in the past decades, as reviewed by Chen [9], Chen et al. [10]. Among various approaches, the Reynolds-averaged Navier-Stokes solver with turbulent model as a closure is still a primary methodology, due its reasonable accuracy and fairly short turn-around computing time [11-14]. However, there are numerous factors that need to be carefully analyzed, such as mesh quality and topology, turbulence model, near wall treatment, among many others, before reliable predictions can be produced and used by the designer.

In this paper, we are going to present a comprehensive validation study of indoor airflows in a 2D empty room with three convection modes; forced convection, natural convection and mixed convection, respectively. The CFD predictions will be compared with the experimental measurements, in terms of velocity and temperature distributions, at the same flow and boundary conditions. Mesh refinement will be carried out, together with turbulence model influence. Finally, modeling of a 3D empty room with forced convection condition will be performed. Time history of flow variables at monitoring points will be analyzed to reveal unsteady flow motions.

II. Computational Methodology

A computer code based on a segregated scheme was used to solve the Reynolds-averaged Navier-Stokes equations in two-dimensional and three-dimensional manner. It is one of the solution algorithms in the Fluent CFD modeling package and by using this approach, the momentum equations are solved separately in sequence in order to update the velocity field. As the resultant velocities may not satisfy the continuity equation on a local control volume, a Poisson-type equation for pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then computed to obtain the necessary corrections to the pressure and velocity fields and the face fluxes to ensure the mass continuity. During the iteration, each discrete governing equation is linearized implicitly with respect to that equation's dependent variable. This results in a linear equation system for each control volume in the computational domain. A point-implicit Gauss-Seidel linear solver is used, in conjunction with an algebraic multi-grid method for solving the matrix equations for each cell. In Fluent, a second-order upwind numerical scheme was chosen for the convection terms treatment and a central scheme for the viscous diffusion terms. The SIMPLE algorithm and its variants were adopted for pressure-velocity coupling. [15]

III. PROBLEM DEFINITION

A. Forced Convection in a 2D Empty Room

Forced convection is a way of transporting heat and energy by an external force via pump, fan, etc. The main features of this flow are that there are virtually no temperature differences between the enclosure walls. Hence due to a relative high momentum of the jet flow, buoyancy effect is small and often be neglected.

There are various experimental cases that can be used for validation and here one configuration studied by Restivo is adopted [16]. As depicted in Fig. 1, a 2D empty room has a height of H and a length of L = 3H. There is a natural air ventilation slot at the front wall near the ceiling with a slot height of h = 0.056H, and at the back wall near the ground, an open outlet with a height of t = 0.16H exists.

A steady air flow at a velocity speed of 0.455m/s is forced at the inlet and it will induce air circulation inside the room. The Reynolds number is 5000, based on the inlet height, the inlet velocity and ambient air conditions, thus flow is expected to have turbulent characteristics. Experiments measured the streamwise velocity (u) profile along the vertical axis at two x-locations of 1H and 2H and previous computational study was also done by Zuo and Chen [17]. Results from present study will be compared with these available data.

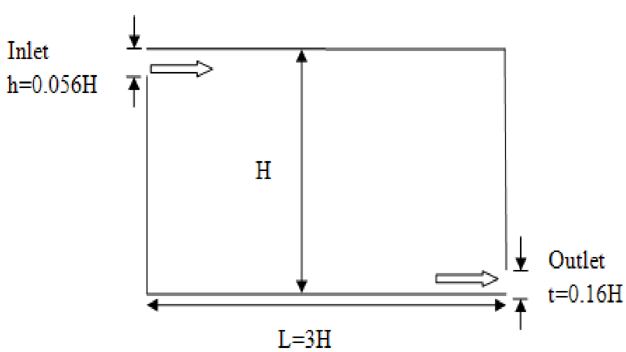


Fig. 1 Geometry of a forced convection flow in a room

B. Natural Convection in a 2D Empty Room

Natural convection is another way of driving the airflow movement by a buoyancy force. In this scenario there is no external force. It is relevant to winter air circulation in a room when all ventilations are closed to preserve the heat. Natural convection of airflow in a tall cavity is a typical example that has been classified as one of basic flow features in buildings.

Among various available cases, an experimental study done by Betts and Bokhari [18] was chosen and this case has been studied numerically by Zuo and Chen [17]. The physical (cavity) domain has a width of 0.076 m and a height of 2.18 m, as shown in Fig. 2. The left wall is cold at a fixed temperature of T_1 = 15.1 ^oC and the right wall is warm at a fixed temperature of T_2 = 34.7 ^oC, while the top and the bottom walls were insulated (i.e. zero wall surface flux), respectively. The corresponding Rayleigh number is 0.86×10^6, which correspondents to a turbulent flow (Betts and Bokhari [18]).

The experiment measured velocity and temperature distributions along horizontal at several different heights from top to bottom. The CFD predictions by Zuo and Chen [17] are also available for comparisons.

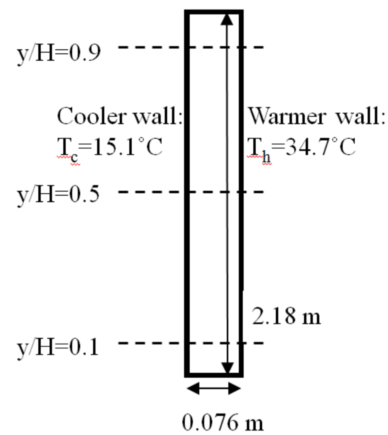


Fig. 2 Geometry of a natural convection flow in a cavity domain

C. Mixed Convection in a 2D Empty Room

The mixed convection represents a combination of forced and natural convections, in which there is an external force introduced near the ceiling, while buoyancy force is not neglegible. One case considering a mixed convective airflow within a sqaure chamber with a heated bottom wall was carried out by Blay et al. experimentally [19], and studied by Chen numerically [20]. Figure 3 dipcits the configuration, in which a jet flow was introduced at the front wall through a slot near the ceiling and an open exit was located at the back wall near the ground. Two side walls and the top wall are kept at a lower constant temperature (15 ^oC) same as that of the inlet jet flow temperautre, while a higher temperature (35 ^oC) is given for the bottom wall, inducing a buoyancy effect. The front and the back walls are treated as adiabatic walls, and top and bottom walls apply fixed temperatures; i.e. isothermal walls.

The experiment was done using a cubic domain, and previous computational study only considered a plane through the center to to make it a pure 2D study, similar approach is adopted in this study.

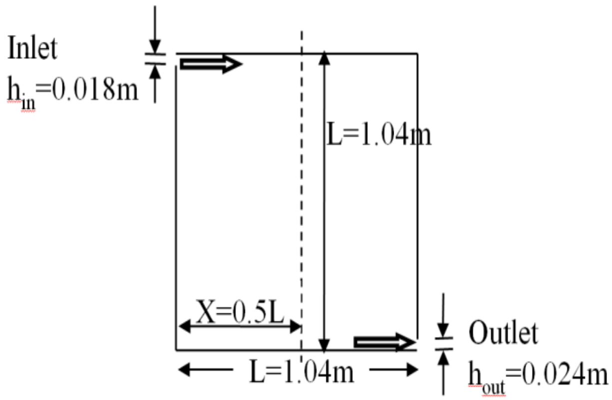


Fig. 3 Geometry of a mixed convection flow in a room

D. Forced Convection in a 3D Empty Room

Forced convection in a 3D empty room is referenced to an experiment described in Zuo and Chen [17]. The purpose of the study was to investigate complex flow features in rooms that represent airflows in an enclosed environment.

A cubic test room of an edge length of 2.44 m was used in the investigation as shown in Fig. 4. There are an inlet slot (at a height of 0.03 m along the whole width) and an exit slot (at a height of 0.08 m along the whole width) at the front and back walls, similar to that of 2D case.

The supply airflow rate was controlled at a value of 0.10m^3/s with a temperature of 22.2 ^oC. There are three scenarios in the experiments with and without a heat boxed inside the room, and here we only consider a scenario in which the room is empty and all walls are insulated; i.e. isothermal conditions. Figure 5 shows a total of ten measurement points in two cross-sections.

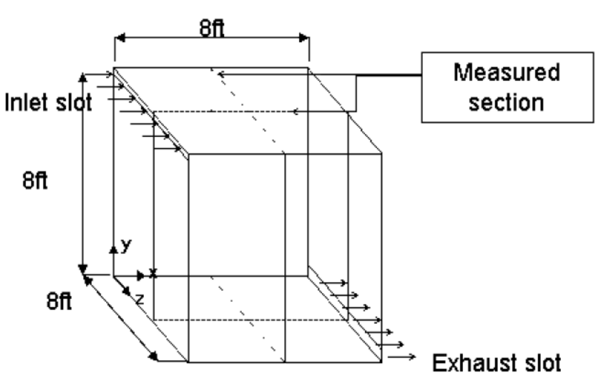


Fig. 4 Geometry of a 3D model with forced convection flow

IV. SOLUTION PROCEDURE

Simulation follows a general strategy as:- first, to start with a mesh in accord to that in relevant literatures, then to consider other mesh topologies to study the effect; second, to carry out grid refinement in each direction; finally, to run a transient simulation in case of noticeable changes in flow variables during the simulation. For steady run, a residual of 1 × 10^{-5} was defined, and for transient run, an estimated physical time, based on fluid particle moving at a convective velocity through a characteristic length (i.e. the edge lengths of the domain).

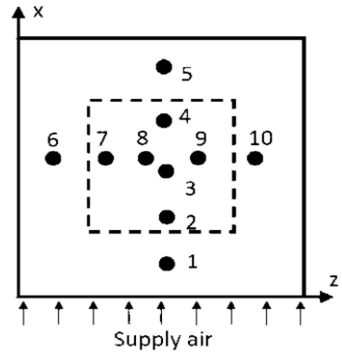


Fig. 5 Measurement locations in experiment [14]

V. RESULTS AND DISCUSSIONS

A. 2D Forced Convection

Simulation starts with a structured mesh of 36 × 36 points, same as that of reference [17], followed by mesh topology, grid refinement studies and transient computations. Results are compared with experimental and numerical data [16, 17] at two streamwise locations x=1H, 2H.

Effect of mesh topology The computational domain is decomposed to three regions; a top region contains inlet jet slot, a bottom region contains outlet open slot and the

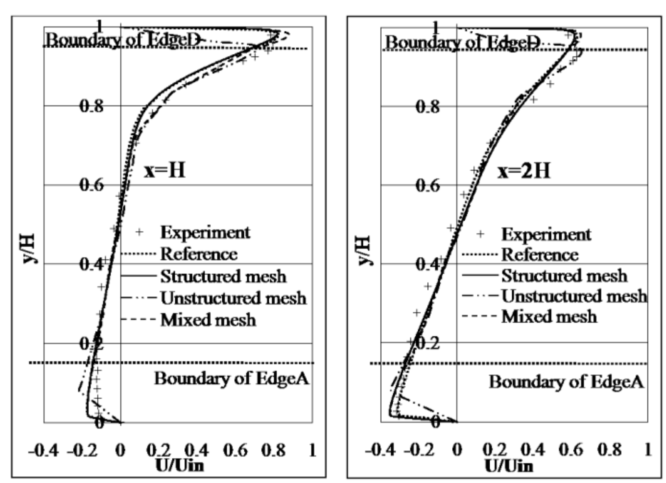


Fig. 6 Effect of mesh topology

remaining middle region. For each region, same grid points as reference [17] were used, and grid size differential at the region interface was minimized by carefully tuning the grid stretch factor in each zone. A mixed mesh uses structured grid in the top and bottom regions, while unstructured grid in the middle region. An unstructured mesh was generated for all three regions. Figure 6 depicts velocity profiles at x=1H, 2H locations. It was found that unstructured grid results give the worst predictions, while both structured and mixed mesh results show good agreement with the measurements and other numerical results. Overall, structured mesh results capture the velocity peak, in better agreement with test data, thus it was decided to apply structured mesh for remaining simulations.

Effect of grid refinements Structured mesh simulation continues by increasing the number of grid points by a factor of 1.5 and 2 in x and y-directions, respectively. Figure 7 shows the x/y-refinement results comparison. It was found that x-refinement has negligible influence, but y-refinement did improve the predictions.

.

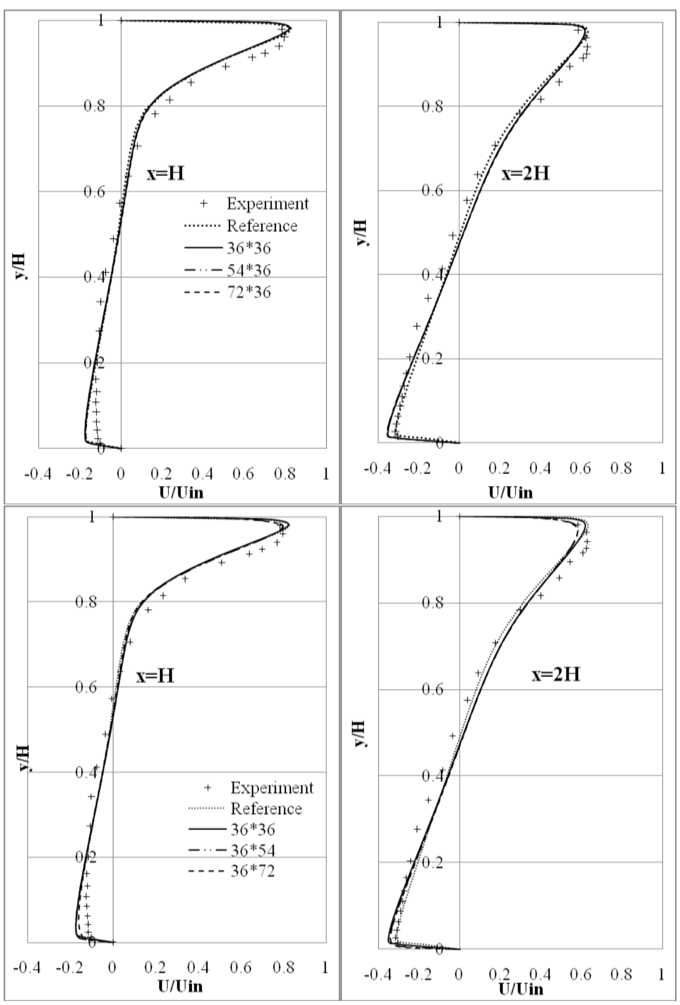


Fig. 7 Effect of grid refinement

Steady and transient simulations Transient simulation was run at a fixed time step of 0.5 seconds, same as that in reference [17]. Comparison of velocity profiles at x=1H and 2H has shown very good agreement and this indicates the airflow movement remains in a steady motion without oscillation, as shown in Fig. 8.

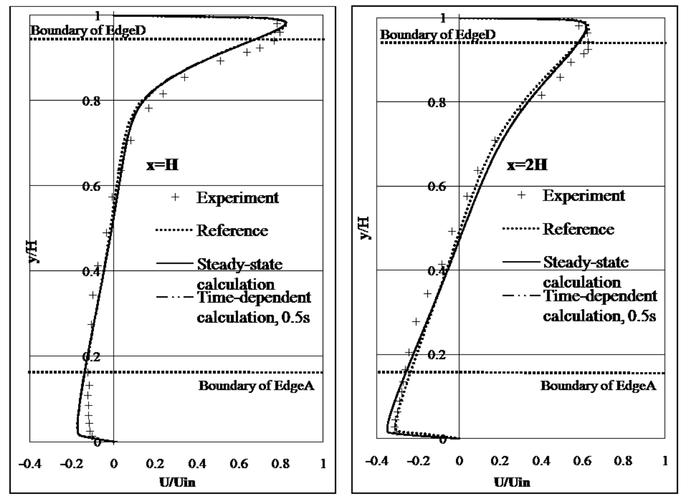


Fig. 8 Comparison of steady and transient runs

B. 2D Natural Convection

Simulation considers steady state run on a structured mesh of 10 × 20 grids, further refined to 20 × 40 grids. Fixed temperatures were given at two side walls and top and bottom walls used adiabatic conditions; i.e. zero heat flux.

Figure 9 shows variation of vertical velocity component in the streamwise direction at three heights of y/H=0.1, 0.5, 0.9, respectively. Buoyancy driven flow and impressible ideal gas assumptions were used, with the former works better at y/H=0.9, while the latter shows better results at y/H=0.1, 0.5.

Figure 10 shows temperature profiles at y/H=0.1, 0.5, 0.9. While present predictions are compared well with the measurement and reference data, it over-predicts at y/H=0.1 and under-predicts at y/H=0.9. Reason for this discrepancy is still not clear yet and further study on this aspect is needed.

C. 2D Mixed Convection

Simulation uses a structured mesh of 20 × 20 grids, same as reference [17], then increases to 30 × 20, and 40 × 20 grids. In addition, different numerical algorithms are used to study their influences on the prediction results.

Effect of algorithm Results from two numerical algorithms SimpleC-GGCB and Simple-GGCB are compared with each

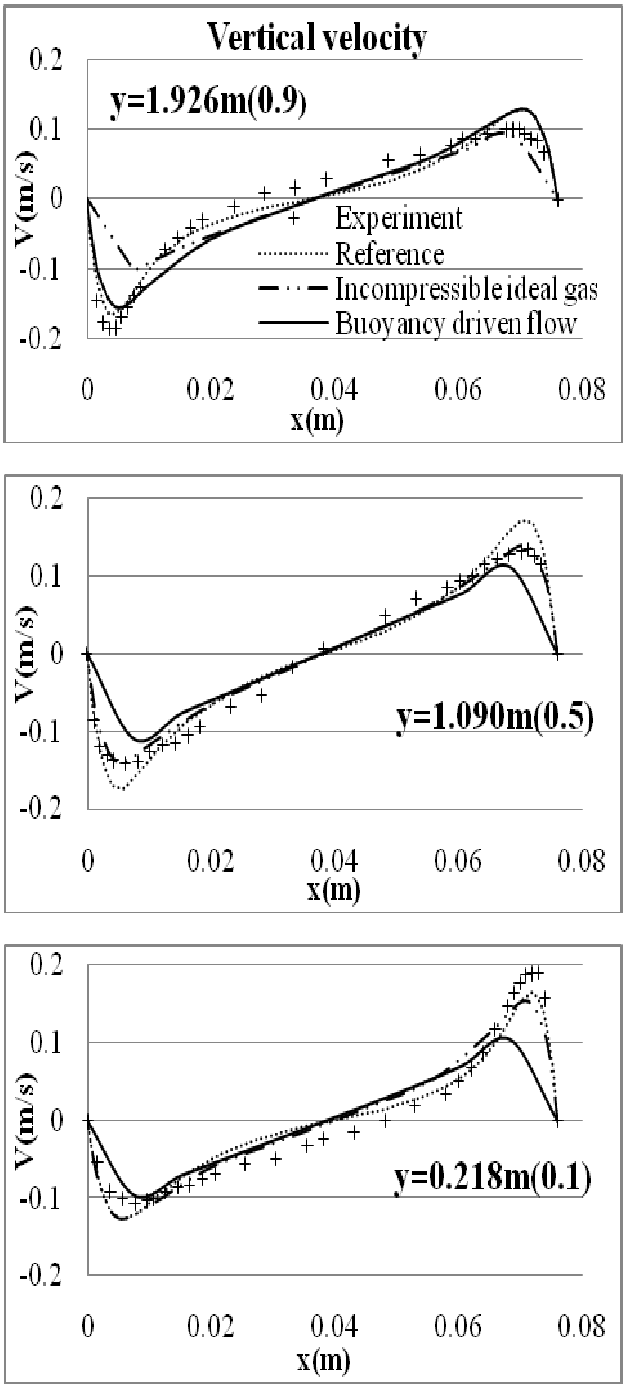


Fig. 9 Velocity profiles at different height

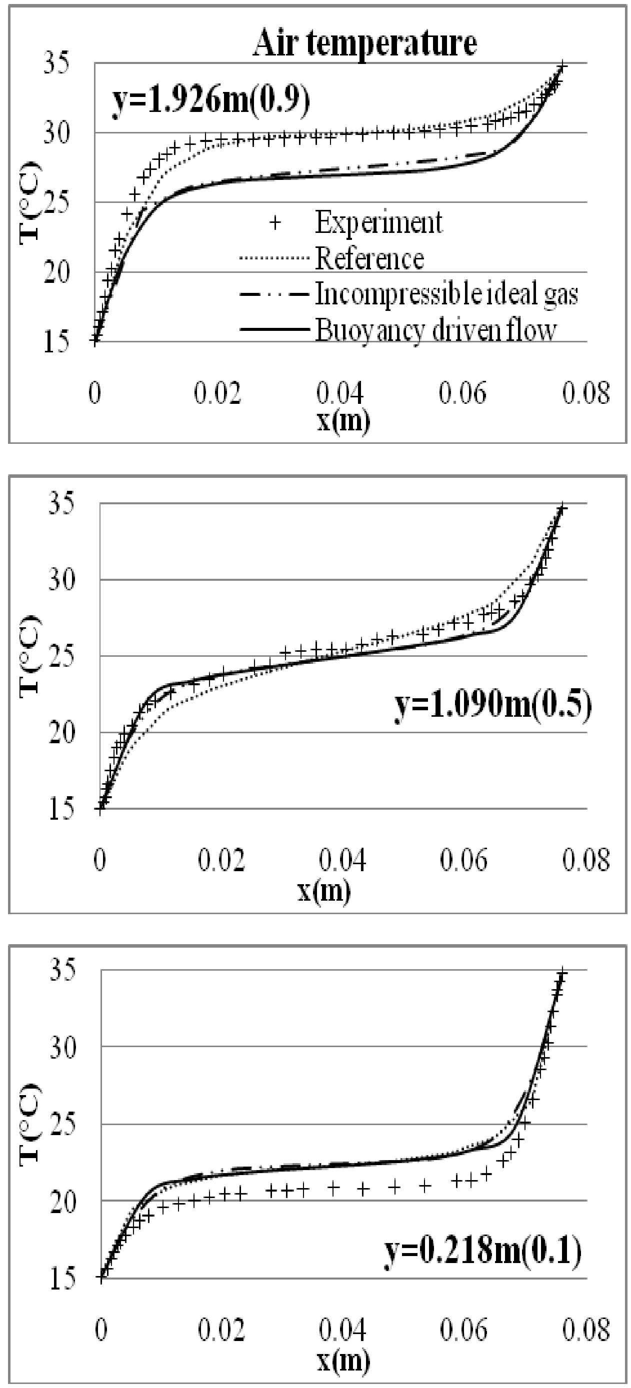


Fig. 10 Temperature profiles at different height

other and also with test and reference data (Fig. 11). The predicted velocity profile is found in better agreement with test data, compared to reference results, while temperature profile agrees well with reference results, both slightly over-predicts in the region near the bottom wall.

Effect of grid refinements Grid refinement in x-direction was carried out and results shown in Fig. 12. The effect on u-velocity component is found insignificant, while predicted temperature on a finer grid did show some improvement, in comparison to test data.

D. 3D Forced Convection

Simulation using a grid of 36 × 36 × 36 was run in a steady mode up to 100,000 iterations with interim results saved per 5,000 iterations and a time history of u-velocity at monitoring

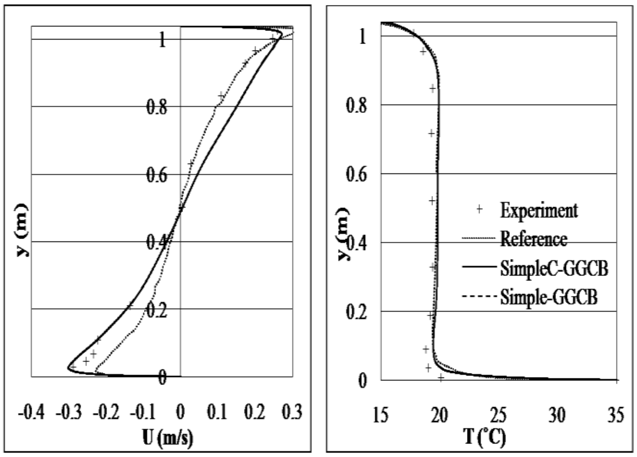


Fig. 11 Effect of numerical algorithm

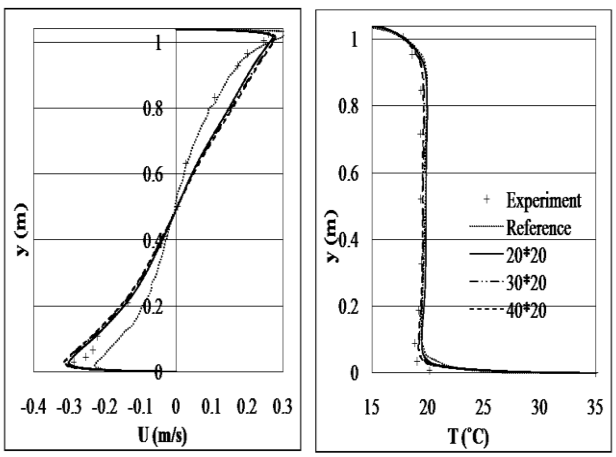


Fig. 12 Effect of grid refinement

positions stored per 20 iterations. Figures 13 and 14 show the time history of normalized u-velocity at measurement points of 5 & 6 with wavy-patterns, indicating flow oscillating at low frequency around its 'mean' position. As a result, the time-averaged 'mean' simulation results were obtained by averaging over 20 interim datasets and compared with reference test data at ten measurement points (Fig. 15). Two turbulence models are considered; i.e. re-normalized group (RNG) k-ε model and shear stress transport (SST) model. It can be seen that results using SST model do not agree well with test data, with significant over- and under-predictions at almost all locations. On the other hand, the RNG k-ε results are overall in good agreement with test data.

VI. CONCLUSIONS

Validation study of airflow in an empty room with three convection modes has been carried out and present results are in good agreement with available experimental measurements and other computational data. While 2D simulation does not

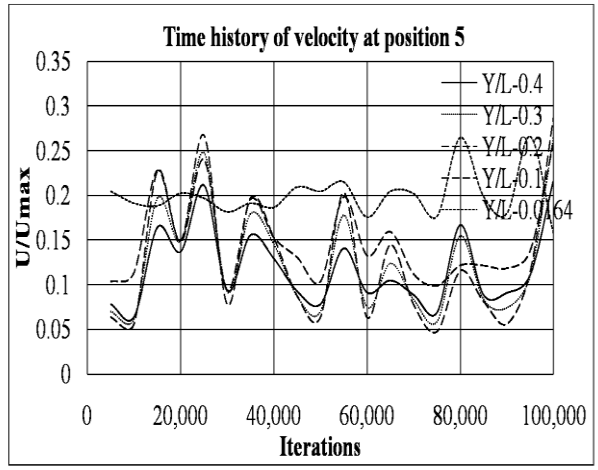


Fig. 13 Time history of u-velocity at position 5

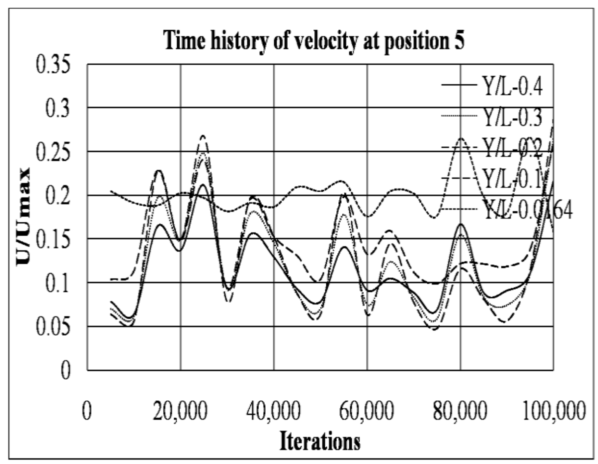


Fig. 14 Time history of u-velocity at position 6

exhibit any flow unsteadiness, 3D simulation does capture oscillating flow features, based on time history of monitored velocity. This kind of flow unsteadiness has not been revealed previously and thus further flow analysis is needed to understand the cause.

ACKNOWLEDGMENT

The first author would like to acknowledge the finance support of Mitsubishi Electric R&D Europe B.V. UK.

REFERENCES

[1] S.D. Fitzgerald and A.W. Woods, "The influence of stacks on flow patterns and stratification associated with natural ventilation," *Building and Environment, 43(10)*, 2008, pp. 1719-1733.

[2] S. Mazumdar and Q. Chen, "A one-dimensional analytical model for airborne contaminant transport in airliner cabins," *Indoor Air, 19(1), 2009, pp. 3-13.*

[3] J.M. Holford and A.W. Woods, “On the thermal buffering of naturally ventilated buildings through internal thermal mass,” *Journal of Fluid Mechanics, 580*, 2007, pp. 3-29.

[4] C.H. Halios and C.G. Helmis, “On the estimation of characteristic indoor air quality parameters using analytical and numerical methods,” *Science of the Total Environment, 380(1-3)*, 2007, pp. 222-232.

[5] C.A. Tatatzikidis, S.A. Piperidis and D.G. Bouris, “Experimental Measurement and 3D Numerical Simulation for the Characterisation of Building Infiltration,” WSEAS Conference of Energy, Environment, Ecosystems, Development and landscape Architecture, 2009, paper no. 22, pp. 151- 156, ISSN: 1790-5095, ISBN: 978-960-474-125-0.

[6] I. Sarbu, C. Sebarchievici, “Simulation and Control of Indoor Air Quality in Building,” WSEAS Advances in Biology, Bioengineering and Environment, 2010, pp. 119-124. ISBN: 978-960-474-261-5

[7] A.F. Najmuddin, I.M. Ibrahim and S.R. Ismail, “Simulation Modeling and Analysis of Multiphase Patient Flow in Obstetrics and Gynecology Department (O&G Department) in Specialist Centre,” WSEAS LATEST TRENDS on APPLIED MATHEMATICS, SIMULATION, MODELLING, 2010, paper no. 19, pp. 125-130. ISSN: 1792-4332, ISBN: 978-960-474-210-3.

[8] T. Spircu, I.M. Carstea, I. Carstea, “Numerical simulation of human thermal comfort in indoor environment,” WSEAS Proceedings of the 3rd WSEAS Int. Conference on FINITE DIFFERENCES - FINITE ELEMENTS - FINITE VOLUMES - BOUNDARY ELEMENTS, 2010, pp. 65-70. ISSN: 1790-2769, ISBN: 978-960-474-180-9.

[9] Q. Chen, “Ventilation performance prediction for buildings: A method overview and recent applications,” *Building and Environment, 44(4), 2009, pp. 848-858*.

[10] Q. Chen, K. Lee, K. Mazumdar, S. Poussou, L. Wang, M. Wang and Z. Zhang, “Ventilation performance prediction for buildings: Model assessment,” *Building and Environment*, 45(2), 2010, pp. 295-303.

[11] V. Yakhot and S.A. Orszag, “Renormalization group analysis of turbulence,” *Journal of Scientific Computing*, 1, 1986, pp. 3-51.

[12] Z.Q. Zhai, W. Zhang, Z. Zhang and Q. Chen, “Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: part 1 - Summary of prevalent turbulence models,” *HVAC&R RESEARCH*, 13 (6), 2007, pp. 853-870.

[13] Z. Zhang, Z.Q. Zhai, W. Zhang and Q. Chen, “Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2- comparison with experimental data from literature,” *HVAC&R RESEARCH,* 13(6), 2007, pp. 871- 886.

[14] M. Wang and Q. Chen, “Assessment of various turbulence models for transitional flow in enclosed environment (RP-1271),” HVAC&R Research, 15(6), 2009, pp. 1099-1119.

[15] ANSYS Fluent user’s manual, *ANSYS* Ltd., 2008.

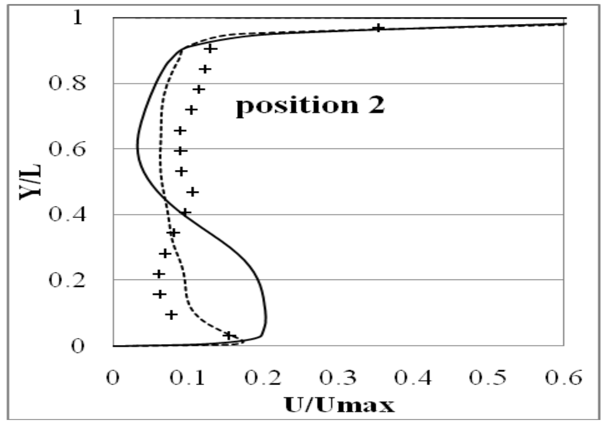
[16] A. Restivo, *Turbulent flow in ventilated room,* Ph.D Thesis, University of London, London, 1979.

[17] W. Zuo and Q. Chen, “Real-time or faster-than-real time simulation of airflow in buildings,” Indoor air,19(1), 2009, pp. 33-44.

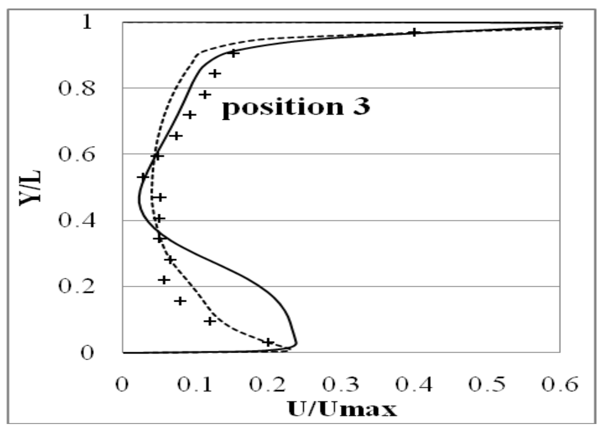
[18] P.L. Betts and I.H. Bokhari, “Experiments on turbulent natural convection in an enclosed tall cavity,” Int. J of Heat Fluid Flow, 21(6), 2000, pp. 675-683.

[19] D. Blay, S. Mergui and C. Niculae, “Confined turbulent mixed convection in the presence of horizontal bupyancy wall jet,” Fundamentals of mixed convection, 213, 1992, pp. 65-72.

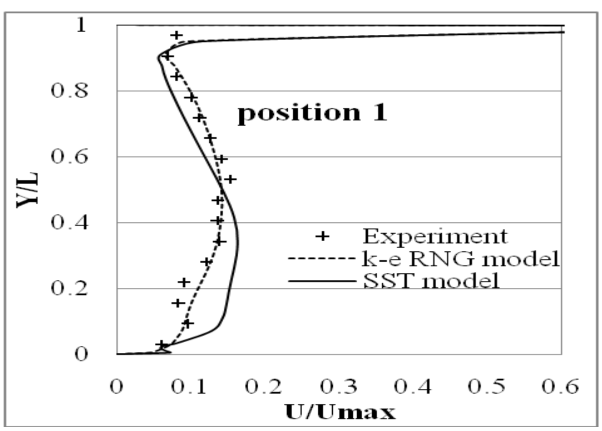
[20] Q. Chen, “Prediction of room air motion by Reynolds-stress models,” Building and environment, 31(3), 1996, pp. 233-244.



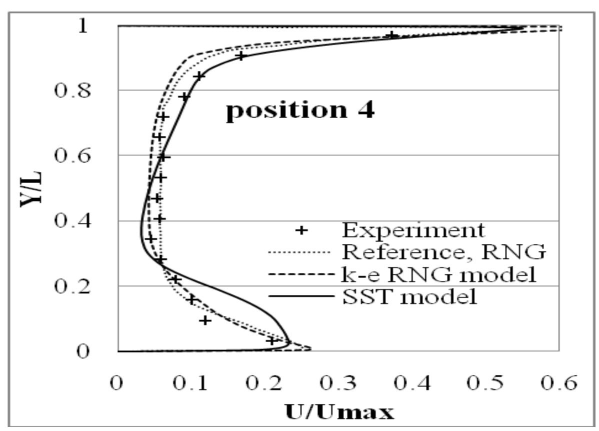
(b)



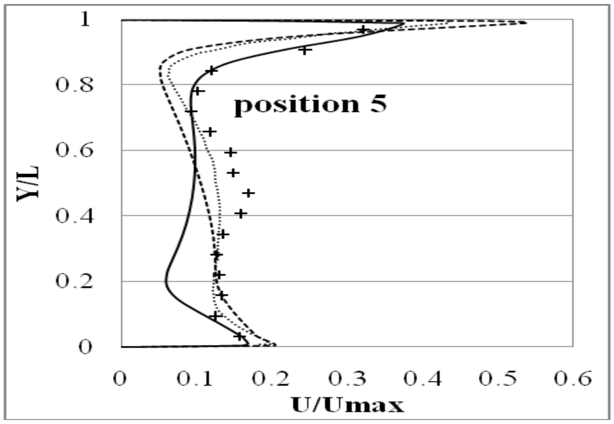
(c)



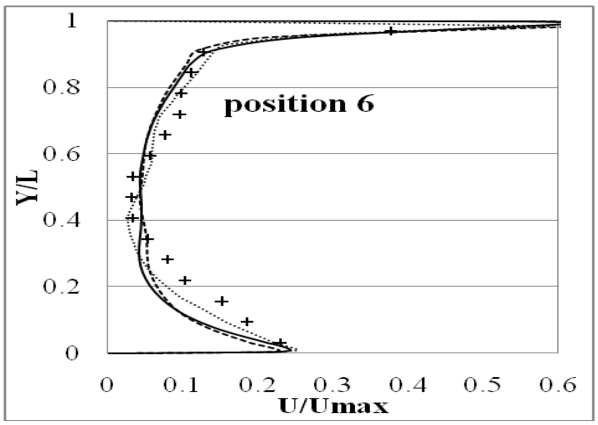
(a)



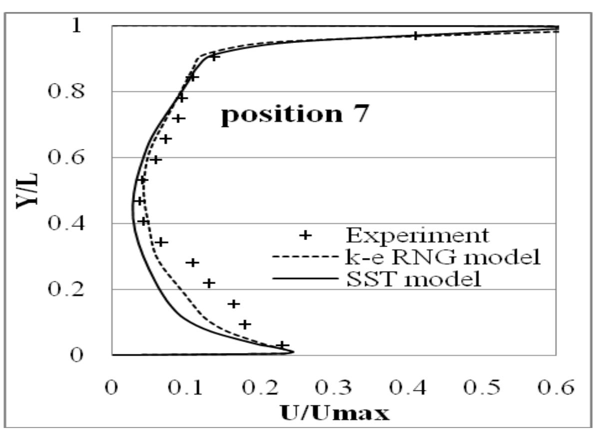
(d)



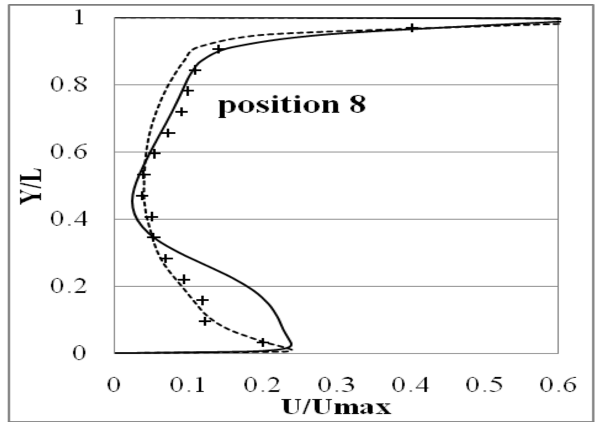
(e)



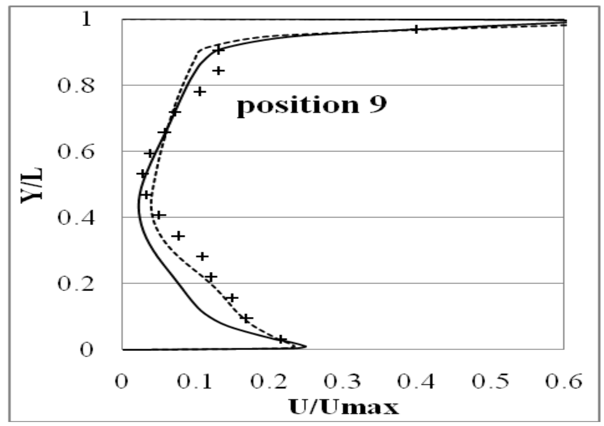
(f)



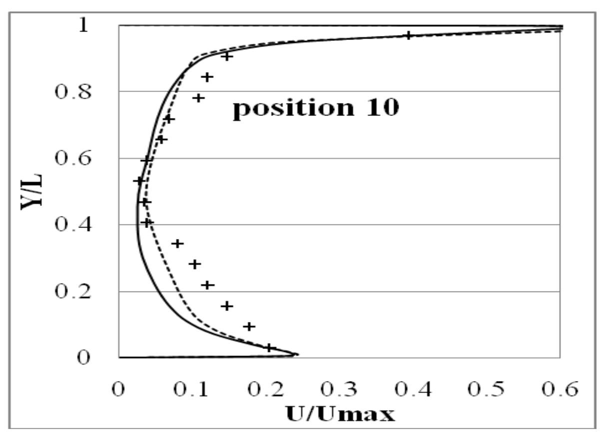
(g)



(h)



(i)



(j)

Fig. 15 Time-averaged ‘mean’ velocity profiles at ten locations in comparison with experiment and reference data