Numerical Modelling of an Indoor Space Conditioned with Low Exergy Cooling Technologies in the Tropics

Esmail SABER[1,2][*], Matthias MAST[1], Kwok Wai THAM[2], and Hansjürg LEIBUNDGUT[3]

[1]Future Cities Laboratory, Singapore, Department of Architecture, ETH Zurich, 8092 Zurich, Switzerland
[2]Department of Building, School of Design and Environment, National University of Singapore, Singapore
[3]Institute of Technology in Architecture, Department of Architecture, ETH Zurich, 8092 Zurich, Switzerland

[*]*Corresponding email:* emsaber@nus.edu.sg

Keywords: low exergy, high temperature cooling, radiant ceiling panel, decentralized air supply units, Computational Fluid Dynamics

SUMMARY

The basic principle of low exergy building systems is the minimization of excess temperature gradients across building systems so that the quality of the energy supplied to the building better matches with the actual demand of the building. Several low exergy technologies which were developed at ETH, Switzerland were implemented in a small research laboratory (BubbleZERO) in Singapore to evaluate the performance of these systems for the tropics. In this paper, the indoor space conditioned by these technologies is analysed by computational fluid dynamics (CFD). Different sources of heat, water vapour and carbon dioxide have been modelled with suitable boundary conditions and emission sources., The CFD results have been verified with different numerical settings and the impact of different turbulence models and mesh densities on the accuracy of the CFD results has been studied. The results have shown that SST k-ω and Reynolds Stress turbulence models have close predictions while the standard k-ε model underestimate the temperature values. In addition, indoor air quality concerns like cold feet or excessive temperature difference for occupants in the space are investigated by checking the thermal stratification in the space.

INTRODUCTION

Most of the countries intend to reduce the greenhouse gas emissions of different sectors in their effort to mitigate global warming. The building sector usually contributes a significant portion of these emissions because of the extensive primary energy consumption of the different services inside the building. The concept of exergy, which is based on the maximum useful work during a process, could be used to supply energy to the buildings in a more efficient way. In a low exergy concept, the heating and cooling demand of the building could be satisfied by a low quality energy source instead of fossil fuel, since the room temperature is usually close to the ambient temperature (IEA ECBCS Annex 49, 2010). This concept can be translated into high temperature cooling and low temperature heating systems for heating, ventilation and air conditioning (HVAC) solutions.

In the tropical context, with high temperatures and humidity levels all around the year, radiant cooling systems, both implemented as panels or integrated with the structure, could be the best representative of low exergy systems for sensible cooling in buildings. Radiant cooling has the potential to satisfy the main part of the space sensible cooling load, while a parallel dedicated outdoor air system (DOAS) handles the latent load of the interior space and the ventilation requirements of its occupants. Moreover, the idea of decentralized DOAS (DDOAS) can further improve the energy and exergy efficiency of the HVAC system by reducing the fan power required to distribute the air in the building. In this design, decentralized units are integrated into the façade and the structure of the building to get the fresh air at the closest location to the façade. The cool and dehumidified air that is brought into the space by decentralized units has to meet the latent load, the ventilation requirement and part of the space's sensible cooling load. The combination of these components is called hybrid DDOAS and radiant cooling system.

The idea of centralized DOAS/radiant cooling panel has been implemented and studied by several research groups for the temperate and subtropical climates (Chiang et al., 2012; Mumma, 2002). Meanwhile, CFD simulation has been extensively used in indoor air studies to study air flow and temperature patterns as well as contaminant dispersion in the indoor space (Li and Nielsen, 2011; Siddiqui et al., 2012). In previous works of the authors (Bruelisauer et al., 2013; Iyengar et al., 2013), the performance of low exergy ventilation technologies in the tropics has been investigated through experimental setup. This study aims to further analyze the performance of the DDOAS/radiant cooling system for the hot and humid climates in terms of the indoor air characteristics by developing and verifying a CFD simulation tool.

METHODOLOGIES

Indoor air space and mesh generation

The ventilation and thermal characteristics of the space are analysed by solving the Navier-Stokes, energy and contaminant transport equations, which account for air flow movement, contaminant dispersion and heat transfer through convection and radiation. The geometry of the decentralized DOAS test bed lab (BubbleZERO) is modelled using the ASNSYS DesignModeller software including the geometry of two seated persons, laptops and other objects in the lab. The BubbleZERO located in an open space with air inflated skin on three sides (North, South, Top) of cube while on the east and west facades there are windows. The details of the modelled indoor air space and real laboratory space are shown in Fig.1. This configuration has been chosen to simulate the typical conditions of the laboratory with two occupants staying inside the space with office activity. In the simulated scenario, two seated users were working with their laptops and the decentralized units were working in parallel with the radiant cooling panel. As can be seen from Fig.1, the fresh air is supplied to the space through seven under floor diffusers and the extracted heat and pollutants leave the space via the exhaust duct, which is integrated with the panels.
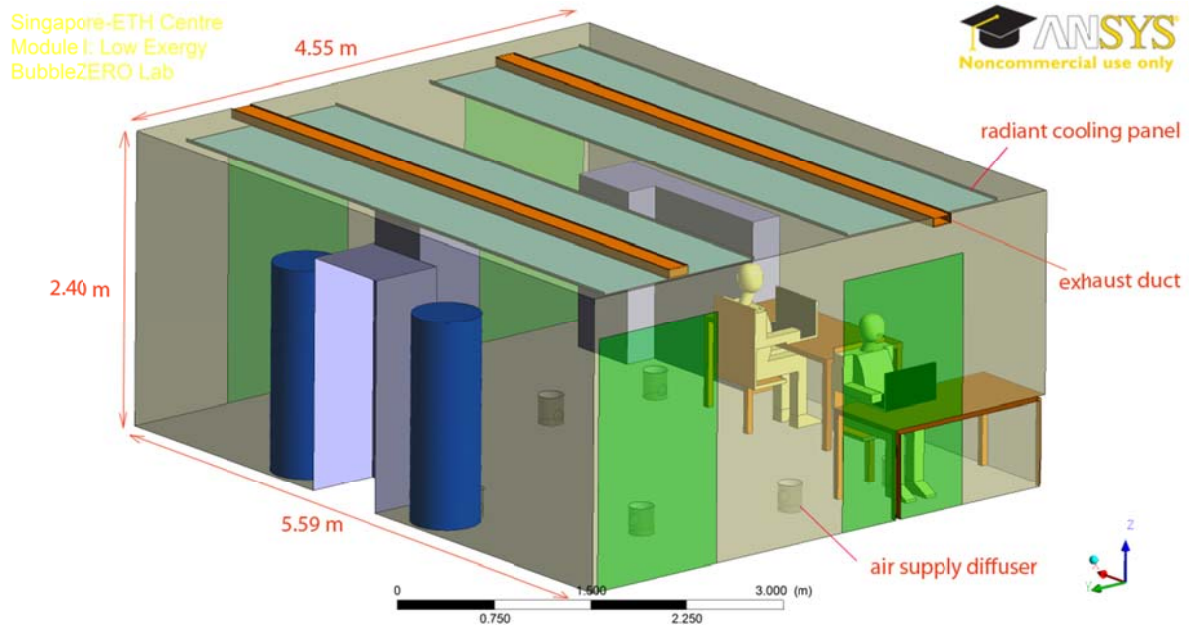
Fig. 1 Geometry of test bed (BubbleZERO) and modelled space for CFD simulation

The complexity of the modelled geometry and the type of mesh volumes in CFD simulations has a significant impact on the accuracy and the solution approach in fluid flow modelling. While a more sophisticated and realistic geometry and higher mesh density could result in more accurate outcomes, it usually requires higher computation time and it also may degrade the quality of the generated meshes and lead to convergence issues. In the case of indoor air flow modelling, the turbulence models, the types of boundary conditions, and the near wall function model are also of considerable importance for achieving a precise distribution. Two different sets of coarse and fine grids have been generated for the indoor space using the ANSYS Meshing program, respectively with 1810 k and 4513 k (Fig. 2) tetrahedron volume meshes.
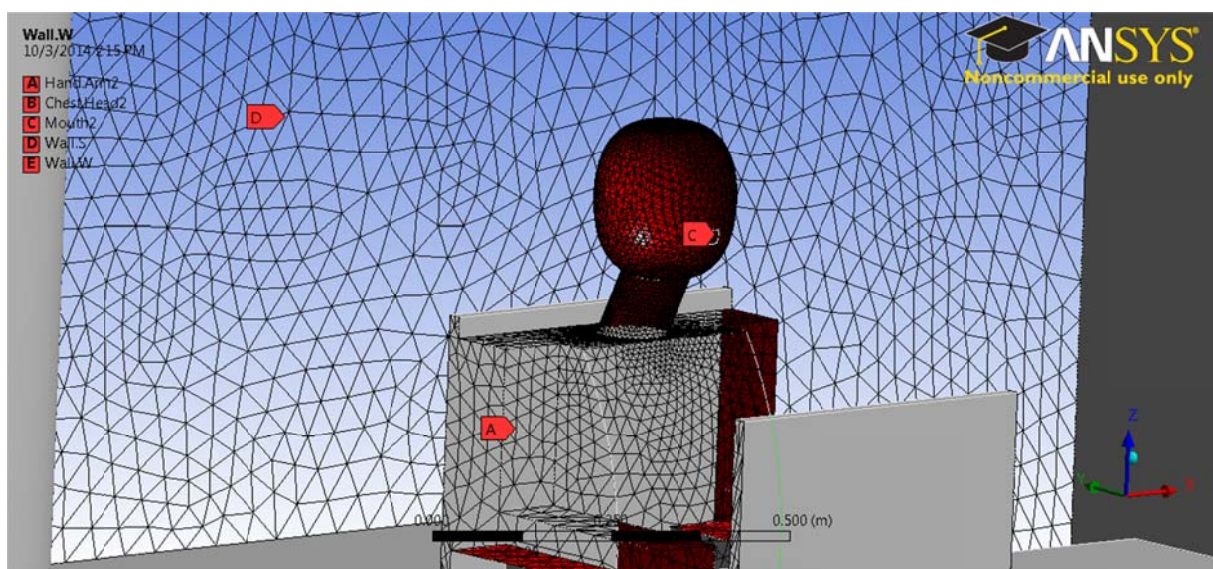

Fig. 2 The generated mesh of the considered geometry consisting of tetrahedrons volume elements

Governing equations and turbulence models

The Navier-Stokes equations including momentum equations based on Newton's law of motion to a fluid element, continuity and energy equations are solved using the commercial CFD packaged of NASYS CFX 14.0. To account for the transport and distribution of water vapour and carbon dioxide in the multicomponent indoor air flow, scalar transport equations are solved for velocity, pressure, temperature and other parameters (mass fraction, turbulent characteristics,…) of each component. Reynolds average Navier-Stokes equations (RANS) are based on the idea of decomposing of variables into time-averaged and fluctuating components and is mostly used for indoor air flow simulation. Compared to other groups of turbulence models like Large Eddy simulation (LES) and Direct Numerical Simulation (DNS), this approach is less computation intensive and more suitable for indoor air CFD studies.

Three different turbulence models have been used to find the most appropriate model for the current problem and to achieve the verified results regarding to uncertainty in numerical solutions. Among standard two-equation models, the k-ε model is the most prominent turbulence model in the industry because of its robustness and stability. It has therefore been implemented in this study. Modelling the accurate behaviour of the air flow near smooth surfaces is a big challenge in turbulence modelling. The k-ω based Shear-Stress-Transport (SST) is designed to give highly accurate predictions of flow patterns in the adverse pressure gradients which happens in the low Reynolds flows near the walls by the inclusion of transport effects into the formulation of the eddy-viscosity. This model has been chosen to see the impact of the k-ω based turbulence models on the accuracy of the results (ANSYS, Inc., 2012b). In flows where the turbulence transport or non-equilibrium effects are important, the eddy-viscosity assumption is no longer valid and solving the Reynolds stresses transport equation is more meaningful. The Reynolds stress model naturally includes the effect of streamline curvature, sudden change in the strain rate, secondary flow and buoyancy (ANSYS, Inc., 2012a). The omega-based Reynolds Stress model (Baseline Reynolds Stress model within ANSYS CFX) with low-Reynolds number formulation has been picked as the third turbulence model. With all three turbulence models, the buoyancy model has been activated to consider the impact of density differences for rising up the warmer air and lighter gas components to the ceiling. In addition, the radiation heat transfer between surfaces has been modelled with the discrete transfer model and grey spectral model.

Boundary conditions and sources

Different sources of heat, humidity and carbon dioxide have been defined in the CFD model to account for the impact of occupants, radiative and convective heat transfer near the walls or through windows. The supply of fresh air through under floor diffusers is modelled with an *inlet velocity* boundary condition based on the typical condition of the supplied air in the afternoon. The walls, windows, floor and ceiling are considered as a constant temperature boundary condition, which represent the steady state heat gain through the structure and facade. The temperature of these surfaces has been set according to the typical temperature of surface in the afternoon. The solar radiation intensity and direction on the windows at the west and east facade has been set based on the computed solar irradiance on a vertical surfaces at 4 pm.

The human impact on the indoor air condition has been considered as a source of heat, humidity and carbon dioxide. An average human produces about 117 W for a typical office

activity which constitutes of 86 W sensible load and 31 W latent load (Stein and McGuinness, 1996). The sensible load of the two modelled occupants is taken into account by assuming constant temperature for different parts of the body which ranges from 28 °C on the foot to 34 °C on the (Yao et al., 2007). Moreover, the latent load of the occupants is taken into account by releasing a constant rate of water vapour through mouth and skin of the body. The total humidity generation of an adult through transpiration and respiration in rest is 50 g/h which one third of this humidity is released through respiration and the rest is through transpiration (TenWolde and Pilon, 2007). And finally, a CO_2 generation rate of 0.005 l/s (0.01 g/s) has been assumed at the breathing point of the occupants based on the releasing rate of an adult male in office activity (Mumma, 2004).

RESULTS AND DISCUSSION

The CFD simulations have been conducted at the high computation centre (HPC) of NUS with a fine and a coarse mesh using different turbulence models. As explained in the Methodology section, the three turbulence models of standard k-ε, SST k-ω and Reynolds Stress have been chosen for this indoor air study. On different computer hosts, the total CPU time ranged between 3 to 5 E+05 seconds for each simulation. In order to avoid convergence issues, the laminar converged solutions of the flow patterns and temperature profile have been chosen as the initial condition for each case. It has been observed in the previously conducted experiments in the lab (Saber et al., 2013) that vertical thermal stratification in the space is in the acceptable range (head to ankle difference < 3 °C) while there are signs of displacement ventilation strategy (ventilation effectiveness > 1). The vertical temperature profiles for the three turbulence models are compared in Fig. 3. The temperature value at each height is the average of the horizontal plane at that location. It can be seen that since the cooling is supplied from both the floor and ceiling, the peak temperature occurs in the breathing zone of the occupants. The temperature at ankle height is around 21 °C and the head to ankle temperature difference is less than 2 °C, which is in the acceptable range (ASHRAE 55, 2004). The standard k-ε profile prediction is slightly lower than the two other models which indicate the weakness of this model on capturing the near wall flow energy diffusion into the space. It is noteworthy that the solution dependent parameter of y plus (average dimensionless distance of the first cell to the walls) is in the order of 1 for the simulations which ascertains the reliability of results.
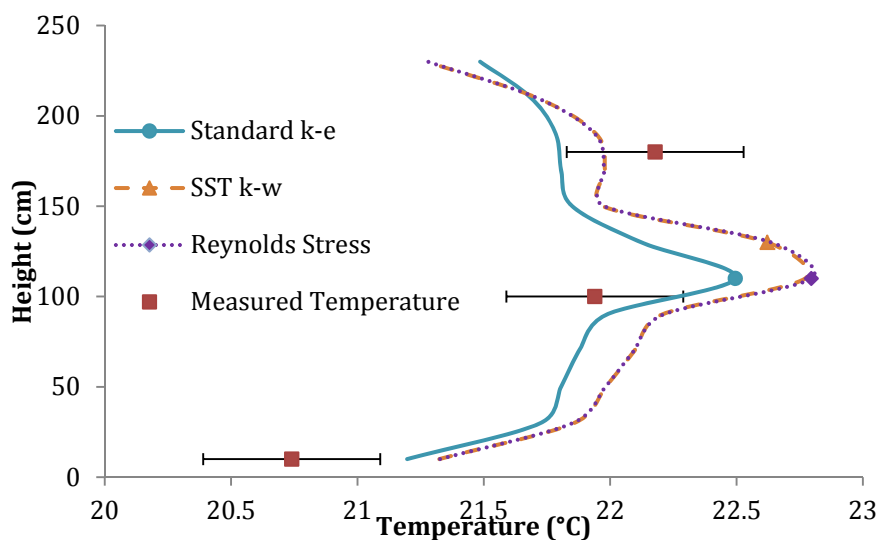

Fig. 3 Vertical temperature profiles in the space using the three turbulence models

The horizontal thermal stratification in the space using coarse and fine mesh grids is plotted in Fig. 4. The temperature value at each length is the average temperature of points on the planes parallel to the width of the space. The maximum temperature fluctuation is around 2 °C on the long side of the space and the fluctuation mostly caused by the position of occupants, underfloor diffusers and windows. In most of the locations, the temperature values for coarse and fine grids are close to each other and the difference is negligible.
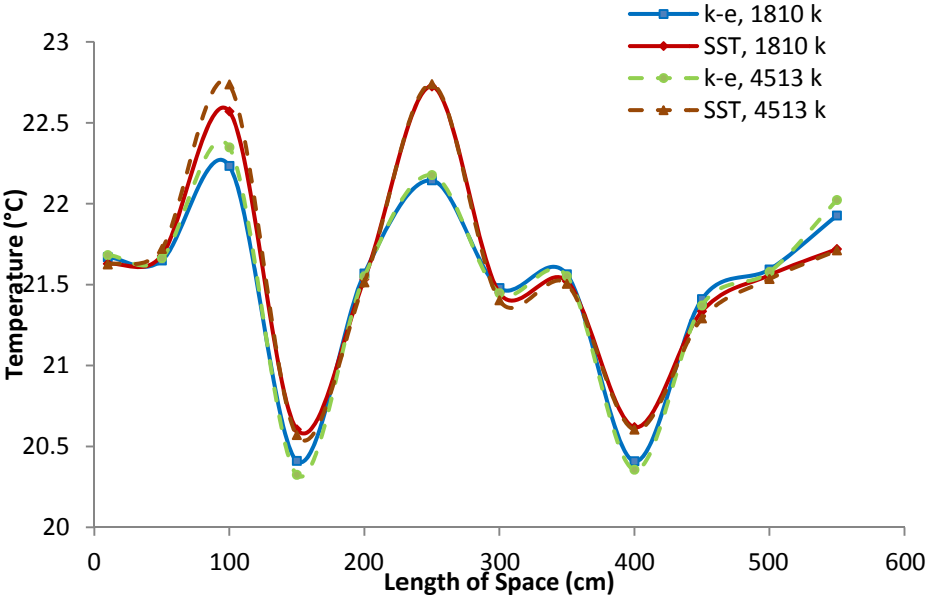

Fig. 4 Horizontal temperature profiles in the space using coarse and fine grids (max values are near the occupants and min values are near the diffusers)

In addition to the study of the thermal stratification in the space, a verified CFD model can be used to investigate the main air streams in the indoor air. Fig. 5 shows the main air streams from the underfloor diffusers to the exhaust ducts on the ceilings. The streamlines show a certain amount of recirculation in the space especially in the left side of the space due to the join of two separate air streams in the opposite directions inside the diffuser. The velocity of air in the space is mostly lower than 0.1 m/s, while inside the inlet diffusers and in the exhaust ducts, the air speed is around 0.2 and 0.9 m/s, respectively.
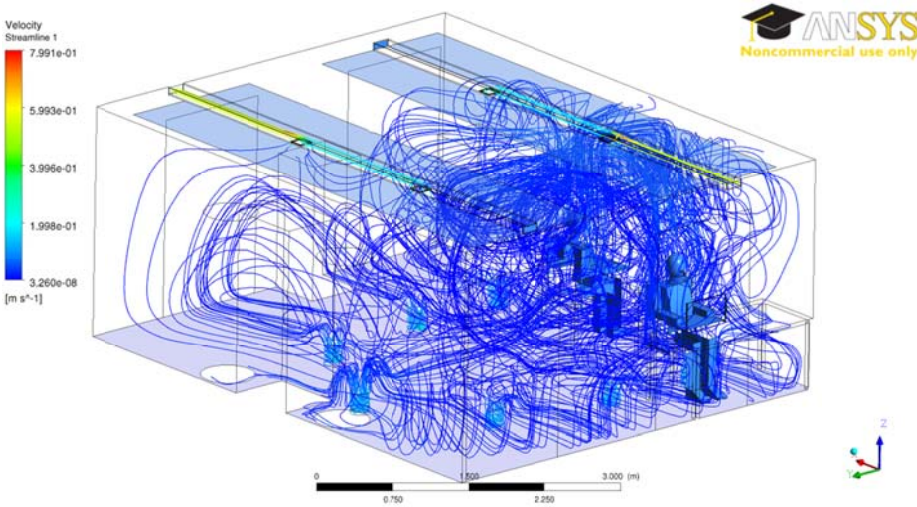

Fig. 5 Main air streamlines from underfloor diffusers to the ceiling exhaust ducts

The contours of temperature and velocity on two planes parallel to the length of the space (one near occupants, one near diffusers) are shown in Fig. 6. The thermal boundary condition and distribution of the air velocity around the human body can be clearly seen in these graphs. Similar patterns of air movement and temperature around the seated body have been reported through numerical and experimental studies in the literature (Sørensen and Voigt, 2003). The other locations with high air speeds include the under floor diffusers and the extract points on the ceiling. The temperature contours also illustrate how the cool supply air spreads in the space. In the region near the supply diffusers, the temperature of around 17 °C raise the concern of cold feet. The distribution of the dew point and carbon dioxide in the space are also determined in these simulations (not shown here). The results reveal that the dew point is around 14-16 °C in most of the locations and that the CO_2 concentration reaches around 700 ppm in the breathing zone of the occupants.

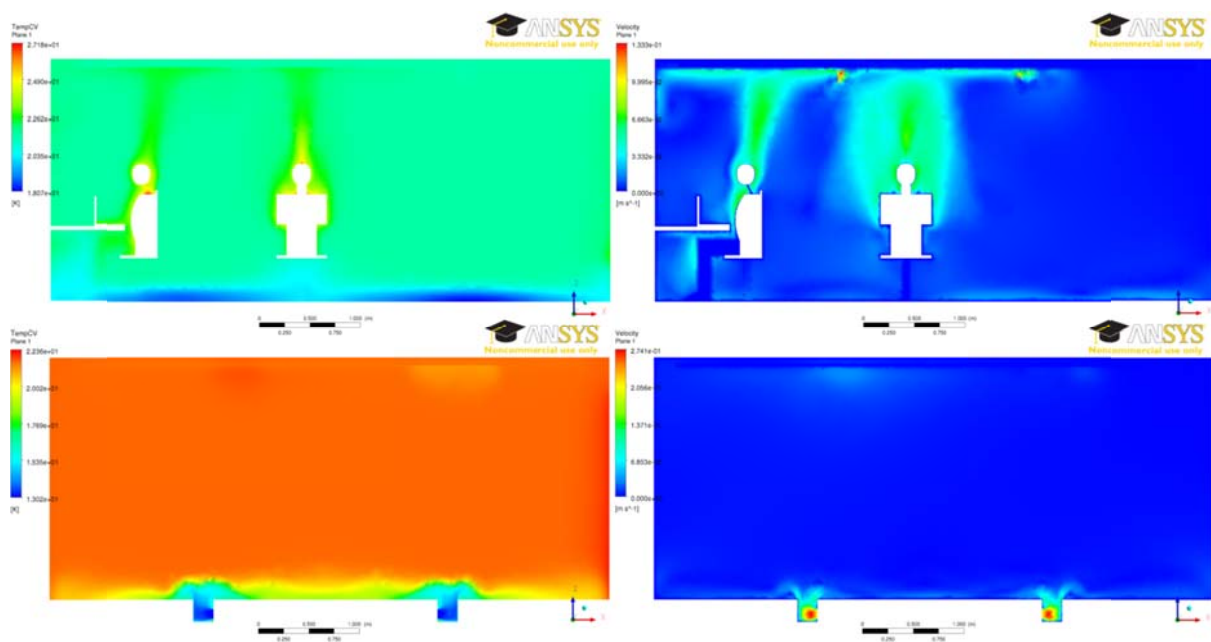

Fig. 6 Contours of temperature (left) and velocity (right) around the occupants and extract openings (top) and around the floor diffusers (bottom)

CONCLUSIONS

CFD simulations have been conducted for the interior space of an air-conditioned lab (BubbleZERO) with hybrid DDOAS (decentralized DOAS) and radiant cooling panels located in the tropical climate of Singapore. The results of the simulations for different turbulence models and mesh densities have been compared to achieve the mesh independent and verified results. The outcomes reveal that SST k-ω and Reynolds Stress models predictions are closer to each other while the standard k-ε model shows lower temperatures at most of the locations in the indoor space . The head to ankle temperature difference and floortemperatures were in the acceptable range based on the ASHRAE 55 standard. However, it has been revealed that the temperature in the area close to the diffusers is around 17 °C which raise the concern of cold feet for those regions. In addition, the predicted dew point level is in the acceptable range of 14-16 °C, which is in the safe mode for the operation of radiant cooling panels without condensation.

ACKNOWLEDGEMENT

REFERENCES

ANSYS, Inc. (2012a) *ANSYS CFX-Solver Modeling Guide*, Canonsburg, PA, SAS IP, Inc.

ANSYS, Inc. (2012b) *ANSYS CFX-Solver Theory Guide*, Canonsburg, PA, SAS IP, Inc.

ASHRAE 55 (2004) *ANSI/ASHRAE Standard 55-2004: Thermal Environmental Conditions for Human Occupancy*, Atlanta, GA, American Society of Heating, Refrigerating and Air-Conditioning Engineers.

Bruelisauer, M., Chen, K., Iyengar, R., Leibundgut, H., Li, C., Li, M., Mast, M., Meggers, F., Miller, C., Rossi, D., Saber, E.M., Tham, K. and Schlueter, A. (2013) BubbleZERO—Design, Construction and Operation of a Transportable Research Laboratory for Low Exergy Building System Evaluation in the Tropics, *Energies*, **6**, 4551–4571.

Chiang, W.-H., Wang, C.-Y. and Huang, J.-S. (2012) Evaluation of cooling ceiling and mechanical ventilation systems on thermal comfort using CFD study in an office for subtropical region, *Build. Environ.*, **48**, 113–127.

IEA ECBCS Annex 49 (2010) *Low Exergy Systems for High-Performance Buildings and Communities*, Frauenhofer Institute for Building Physics, Available from: http://www.annex49.com.

Iyengar, R.S., Saber, E., Meggers, F. and Leibundgut, H. (2013) The feasibility of performing high-temperature radiant cooling in tropical buildings when coupled with a decentralized ventilation system, *HVACR Res.*, **19**, 992–1000.

Li, Y. and Nielsen, P.V. (2011) CFD and ventilation research, *Indoor Air*, **21**, 442–453.

Mumma, S.A. (2002) Radiant Cooling Panel Systems, Available from: http://doas-radiant.psu.edu/IAQ5.pdf (accessed 17 August 2013).

Mumma, S.A. (2004) Transient occupancy ventilation by monitoring CO2, *ASHRAE IAQ Appl.*, 21–23.

Saber, E., Meggers, F. and Iyengar, R. (2013) The potential of low exergy building systems in the tropics - Prototype evaluation from the BubbleZERO in Singapore. In: *Proceedings of Clima 2013: Energy efficient, smart and healthy buildings*, Prague, Czech Republic.

Siddiqui, M., Jayanti, S. and Swaminathan, T. (2012) CFD analysis of dense gas dispersion in indoor environment for risk assessment and risk mitigation, *J. Hazard. Mater.*, **209-210**, 177–185.

Sørensen, D.N. and Voigt, L.K. (2003) Modelling flow and heat transfer around a seated human body by computational fluid dynamics, *Build. Environ.*, **38**, 753–762.

Stein, B. and McGuinness, W.J. (1996) *Building technology: mechanical and electrical systems*, New York, J. Wiley.

TenWolde, A. and Pilon, C.L. (2007) The effect of indoor humidity on water vapor release in homes. In: *ASHRAE thermal performance of the exterior envelopes of whole buildings X*, Atlanta, Ga., American Society for Heating, Refrigerating, and Air-Conditioning Engineers.

Yao, Y., Lian, Z., Liu, W. and Shen, Q. (2007) Experimental Study on Skin Temperature and Thermal Comfort of the Human Body in a Recumbent Posture under Uniform Thermal Environments, *Indoor Built Environ.*, **16**, 505–518.