Proceedings of the 13th Asian Congress of Fluid Mechanics
17-21 December 2010, Dhaka, Bangladesh

Analysis of Building Systems Performance through Integrated Computation Fluid Dynamics Technique

Ashfaque Ahmed Chowdhury[1*], M G Rasul[2], M M K Khan[2]

[1]Process Engineering and Light Metal Centre, School of Engineering and Built Environment, Faculty of Sciences, Engineering and Health, CQ University, Gladstone, Qld 4680, Australia.
[2] School of Engineering and Built Environment, Faculty of Sciences, Engineering and Health, CQ University, Rockhampton, Qld 4701, Australia.

*E-mail of presenting author: a.chowdhury@cqu.edu.au

Abstract Computational fluid dynamics (CFD) has immense effect in building system performance. There have been few studies on the potential use of CFD in building concerning the evaluation of external and internal flow. In this study, CFD module of DesignBuilder (state of the building energy simulation software) has been designed to predict airflow and temperature distribution in buildings. Conventional CFD packages (FLUENT, PHOENICS, CLIMA 3D etc.) for building airflow analysis is time consuming and intensive task to setting up the correct geometry and boundary conditions. In this study, a simplified simulation method has been followed by providing the geometry and boundary conditions in the CFD module of Design Builder. Calculated temperatures, heat flows and flow rates of air from the building system have been provided boundary conditions simply by specifying the time/date of the CFD analysis. The study employs SIMPLER algorithm, which belongs to one of the most widely used families of CFD solution methods. Turbulence is modelled using standard k-e model. A wide range of boundary conditions such as supply diffusers, extracts, temperature patches, etc. have been assigned to room surfaces. Component library has been used to include building systems (radiators, fan-coil units) and occupants within the analysis. Using time dependent CFD simulation, it is possible to control time dependent thermal comfort by changing the boundary condition. In this study, CFD simulation integrated with whole building energy model will be used to obtain more accurate prediction of the building performance.

Keywords Building Systems Performance, Computational Fluid Dynamics (CFD)

1. Introduction

Computational fluid dynamics (CFD) has immense effect in building system performance. There have been many studies on the potential use of CFD in building concerning the evaluation of external and internal flow. Zhai et al [1], Bartak et al. [2] and Djuneady et al. [3] have tried to combine CFD with the analysis method of total building energy. The major applications of CFD in internal flow are related with MVAC system performance and prediction of ventilation rate. Muller and Renz [4] evaluated the performance of the different turbulence model in room airflow applications using K-e model, low renolds model and RSM. Hagstrom et al [5] describe the different air distribution methods of room air conditioner and compare the heat, humidity etc. Holmberg et al [6] investigated indoor air quality and climate control parameters by CFD analysis. Karimpanah et al [7] tested four different air distribution systems with realistic load to predict air temperature, air velocity, ventilation effectiveness etc. and developed CFD simulation. Zhai et. al. [8] coupled different strategies of energy simulation program (Energy Plus) and a CFD program (MIT-CFD).Combining CFD and energy simulation has the advantage of better airflow.

CFD module of DesignBuilder has been designed to predict airflow and temperature distribution in and around buildings using the same methods as the general CFD packages. Conventional CFD packages (FLUENT, PHOENICS, CLIMA 3D etc.) for building airflow analysis is time consuming and intensive task to setting up the correct geometry and boundary conditions. CFD module of DesignBuilder simplifies this process hugely by automatically providing the geometry and boundary conditions. Temperatures, heat flows and flow rates calculated by EnergyPlus can be seamlessly used to provide boundary conditions simply by specifying the time/date of the CFD analysis. CFD interface of DesignBuilder enable virtual CFD analyses from automatically generated rule-based default data.3D CFD grids are automatically generated from model geometry and boundary conditions and can be modified to promote solution convergence. The CFD engine employ SIMPLER algorithm, which belongs to one of the most widely used families of CFD solution methods. Turbulence is modelled using k-e model. The interface incorporates tools to enable a wide range of boundary conditions such as supply diffusers, extracts, temperature patches, etc. to be assigned to room surfaces. In this study, a building systems performance have been simulated and thermal comfortability of the occupants have been simulated using computational fluid dynamics techniques.

2. Weather Condition

Rockhampton weather is considered as Subtropical. The city is situated in the Capricorn region and lies within the southeast trade wind belt. Rockhampton's rainfall average classified the city as a distinct wet and dry season, with the wet season generally December to March and the dry season June to September. Generally, summer is from December to February and winter is from June to August. For the simulation, extreme hot summer period was selected from January 27 to February 2 and nearest maximum summer temperature was taken 39°C. In typical summer week, the nearest average temperature is 26.38°C. Extreme cold winter week is selected from June 8 to June 14 and nearest minimum temperature for winter is 5.00°C. In typical winter week the nearest average temperature is 16.99°C.

3. Model Description

The modelled conventional residential house is made of concrete slab and/or footings for foundation, external walls, internal walls, timber roof structure with tiles or colour bond. External wall consists of external 110 mm brick veneer, 50 mm air cavity, thin isolation foil, approximately 90 mm timber structure filled with insulation batts and 10 mm plaster board. Internal wall is generally comprised approximately 90 mm timber structure and 10 mm plasterboard on both sides. The house has only one level and has a complete air conditioned area. A model case with the conventional settings has been developed in DesignBuilder platform [Fig. 1].

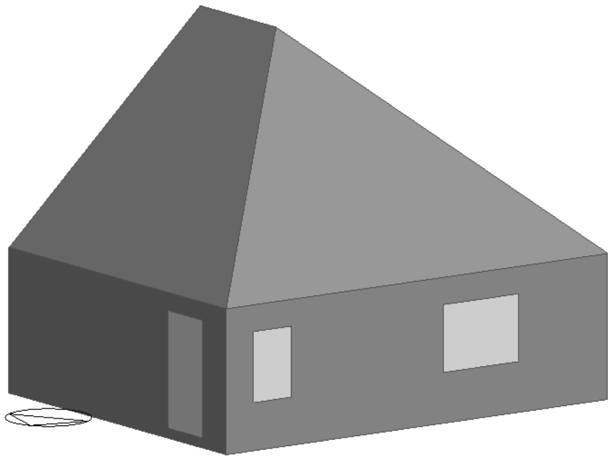


Fig 1: Schematic Diagram of the house considered in the simulation

The boundary conditions used for the simulation are listed below

Operating Schedule: 8:00 to 18:00 [5 days/week]
Occupancy: one person per $10m^2$
Outside air rate: 10L/s/person
Lighting Type: Compact fluorescent
Lighting Power Density: $18w/m^2$
Office Equipment Power Density $15w/m^2$
Cooling Type: Air Cooled
Cooling Power Density $40w/m^2$
Ventilation Power Density $5w/m^2$

4. Simulation Principles

In this study, DesignBuilder [9] is used to assess thermal comfort ability in a residential house located in Rockhampton, Australia. Current version of DesignBuilder (DB) allows EnergyPlus (EP) [10] as the calculation method to evaluate the energy performance of the building. DB creates a virtual environment where building systems are evaluated. Residential house systems analysis based on simulation as well as climate analysis are presented. The CFD technique is based on solving a set of partial differential equations describing a set of partial differential equations involving transport of momentum, energy and turbulence quantities. The concept of CFD simulation is shown in Fig 2.

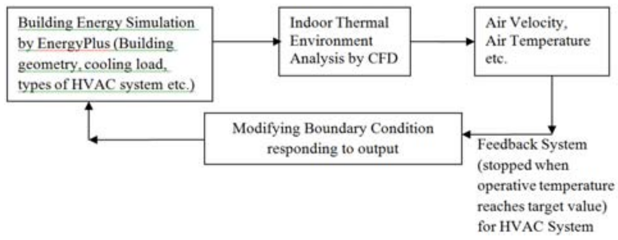


Fig 2: Concept of CFD Simulation for Building Systems

A great strength of CFD technique is the clear and accessible output formats: color contours, vectors and streamlines, which can easily be understood. The grid is arranged to fit to the geometry of objects in the model building as shown in Fig 1. The number of cells is 28 x 23 x 22. Because the grid is a discrete representation of the continuous field phenomena, the accuracy and numerical stability of simulation depend on the choice of grid. In this CFD study, the standard k- ε model was used for turbulence modeling. The assumption made in the simulation is that the airflow is incompressible and steady-state in the air-conditioning system in the buildings. The governing equations for the general dependent variable, φ, can be expressed by the following general equation:

$$\frac{\partial}{\partial t}(\rho\phi) + div(\rho u\phi) = div(\Gamma grad\phi) + S$$

where ρ is the density, Sφ the source/sink rate per unit volume for the dependent variable φ, and Γ φ is the effective exchange coefficient of φ.

The computational procedure adopted for such a three-dimensional turbulent airflow is based on solving the governing equations for the dependent variables (velocity components in x, y and z direction, and the pressure) by means of the finite volume technique. In the simulation, the velocity component is checked at the inlet and outlet planes to ensure that the flow rate has a constant value. DesignBuilder has an automatic, unstructured hybrid element mesh generator with an adaptive mesh refinement algorithm to allow a very accurate representation of the boundaries. Grids were used to consider locations with large gradients of the solution variables (e.g. air velocity, temperature). Grid refinement occurs on the surface of heat sources and in the inlet and outlet regions. In the boundary definition, the airflow rates and temperatures of the air supply diffusers and jets were set equal to their actual values. The flow was assumed to be uniformly distributed on all these supply openings with a constant vertical velocity. The boundary condition of temperature was specified in the model to represent the temperatures on the interior surface of the surrounding walls, floor and ceiling. Heat fluxes were modeled to represent the actual amount of heat generated by the types of heat sources.

5. Results and Analysis

Heating and cooling design calculations are carried out using simple worst-case winter and summer design data from ASHRAE to determine the size of the heating and cooling equipment required to meet the coldest and hottest winter and summer condition likely to be in Rockhampton. Heating and cooling design calculations are done by putting a sin curve through maximum daytime and corresponding night-time summertime design temperatures. By default, EnergyPlus assumes that air temperature within a zone is completely uniform (i.e. the air is fully mixed). Heating design calculation predicted the heat loss of each zone of the building at steady state with no solar gain. This is based on worst-case winter design data for the location of the building. The cooling design calculations are based on simplified sinusoidal worst-case summer design conditions. It is found that the average air temperature of the air-conditioned spaces of the house is within the range of twenty which also meets the comfort temperate range (Fig. 3).

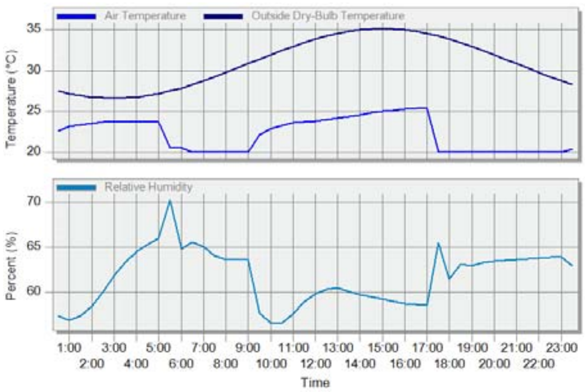


Fig 3: Simulated indoor air temperature and humidity.

According to ambient atmospheric pressure, the inlet and outlet pressures were assigned to be zero. The ambient temperatures were simulated. Velocity profiles are important for the visualisation of the building performance. Simulations have been performed for a flow rate 10 l/s to investigate the air flow in the model house. Fig 4 shows the magnitude of velocity vectors and found within 0.01 m/s to 0.07 m/s with little variation in the inlet and outlet zone (windows and doors). The velocity vectors appear to change gradually to nearly uniform flow as the flow moves towards the doors and windows.

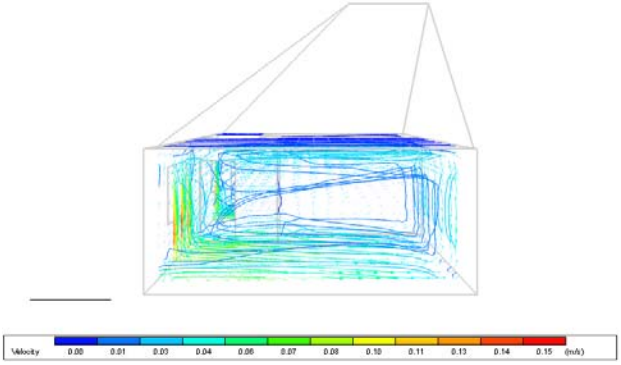


Fig 4: Distribution of simulated velocity contours on the x-y plane

Fig. 5 illustrates the simulated temperature contours. It can be observed that the temperature vectors are rapidly changed to nearly uniform flow as the flow passes throughout the inlet zone. The indoor temperatures within the air-conditioned spaces are also found within the 20^{o}C range through the house. The Fig. 6 shows the air velocity and temperature distributions for the CFD simulation results. In order to determine the air velocity and temperature performance were chosen for determining the comfort level of the occupants. The results suggest that indoor CFD simulations using pressure-inlet boundary conditions give better predictions than using velocity-inlet boundary conditions.

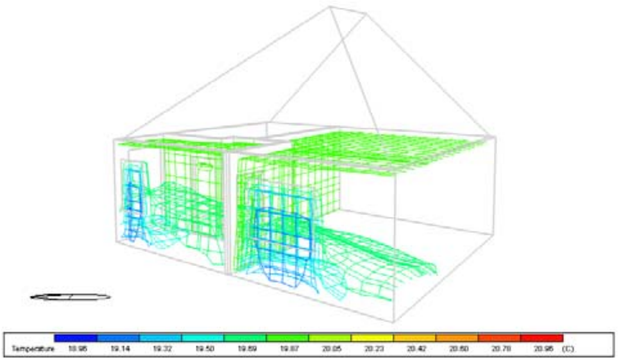


Fig 5: Distribution of simulated temperature contours

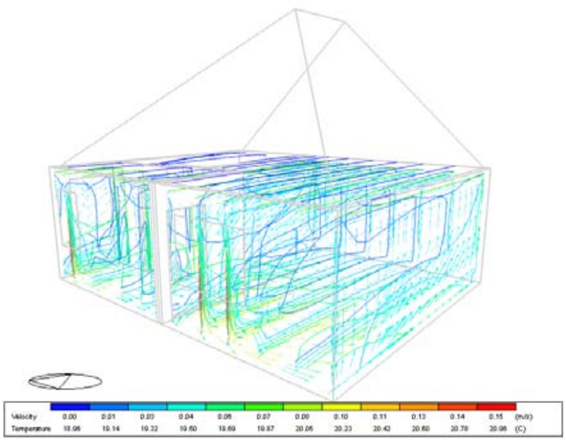
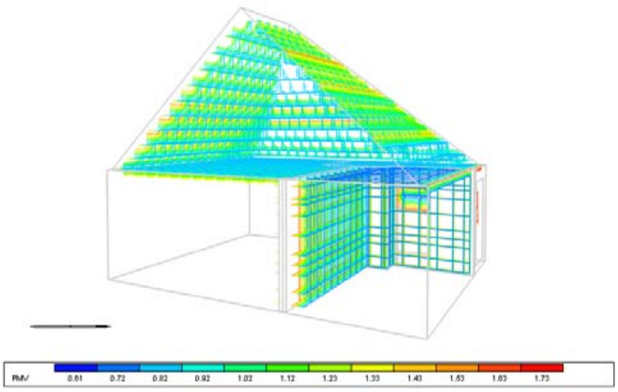
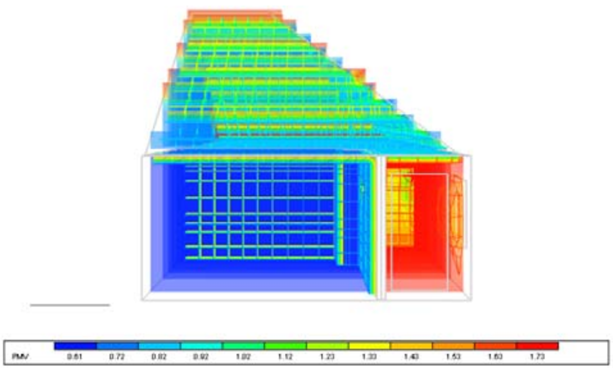


Fig 6: Distribution of velocity and temperature contours

In this study, extensive data on environmental conditions within the building and occupants' comfort level have been studied. The temperature and humidity level are found consistent and maintained considerably within the comfort level. Through the simulation, the current indoor environmental control strategies have been checked and thermal comfort ability of the building has been determined. Thermal comfort index (Fanger PMV) has also been simulated on nine point thermal sensational scale and results are with in ±1 where +1 stands for slightly warm and -1 stands for slightly cool. The result of the Fanger PMV simulation is plotted in Fig 7. For most of the living rooms and roof space, the thermal comfort level is found within the $-1.0 < PMV < +1.0$ limits for 10% PPD as per ISO 7730-1994 [8] during office hours on summer and winter days. The only exception is the kitchen (Fig. 7 b) where the thermal comfort limit is relatively high (right right) due to the nature of the activities.



(a)



(b)

Fig 8: Distribution of filled (a) PMV slice contours (b) filled PMV contours

6. Conclusions

The CFD technique is used to analyse building systems performance in order to evaluate the indoor thermal environment of an air-conditioned residential house. The simulated demonstrates the satisfactory uniform velocity and temperature distributions in the occupied region of the house. The results also indicated that analyzing the thermal comfort in the built environment with CFD is an effective method to find the way to determine the comfortabiliy of the occupants. The thermal CFD simulation model describes in this study incorporates the latest turbulence modelling advancements applicable for room air flow simulation and resolves the room's air flow and temperature distribution, and to predict surface convection. It is found that the simulation outputs can give productive and convincing numerical details of the actual thermal comfort on the building occupants. Moreover, the simulation results appear to be adequate for the practical applications. CFD Simulations results of the whole building show that the thermal comfort level is maintain fully and in most cases it is within comfort zone.

References

[1] Zhai, Z., Chen, Q., Klems, J. H., & Haves, P., "Strategies for coupling energy simulation and computational fluid dynamics programs", CA: Lawrence Berkeley National Laboratory. 2001.

[2] Bartak, M., Beausoleil-Morrison, I., Clarke, J. A., Denev, J., Drkal, F., Lain, M., et al., "Integrating CFD and building simulation", *Building and Environment*, 37 (8-9), 865-871, 2002.

[3] Djunaedy, E., Hensen, J. L., and Loomans, M. G., "Towards external coupling of building energy and air flow modeling programs", *ASHRAE Transactions*, 109 (2), 771–787, 2003.

[4] Muller, D., & Renz, U.. "Measurements and predictions of room airflow patterns using different turbulence models", Proceedings of the Roomvent'98. Stockholm, Sweden: *6th International Conference on Air Distribution in Rooms*, 1998.

[5] Hagstrom, K., E, S., Koskela, H., & Hautalampi, T., "Room Air Conditioning Strategy". Halton Group, 2002.

[6] Holmberg, S., Sandberg, M., Mattsson, M., Nilsson, M., & Holmer, H., "Indoor air quality and climate control parameters in an office environment CFD calculations and measurements," Proceedings of the Roomvent 2000. Reading, UK: *7th International Conference on Air Distribution in Rooms*, 2000

[7] Karimipanah, T., Sandberg, M., & Awbi, H. B., "A comparative study of different air distribution systems in a classroom", Proceedings of the Roomvent 2000, Reading, UK: *7th International Conference on Air Distribution in Rooms*, 2000.

[8] Zhai, Z., Chen, Q., Haves, P., & Klems, J. H., "On approaches to couple energy simulation and computational fluid dynamics programs", *Building and Environment*, 37 (8-9), 857-864., 2003.

[9] DesignBuilder User Manual 2006, Version 1.2, DesignBuilder Software Limited, UK.

[10] EnergyPlus Manual 2006, Documentation Version 1.4.