Available online at www.sciencedirect.com
ScienceDirect
ELSEVIER
Building and Environment 44 (2009) 95–112
BUILDING AND ENVIRONMENT
www.elsevier.com/locate/buildenv

Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment

Liping Wang, Nyuk Hien Wong*

Department of Building, School of Design and Environment, National University of Singapore, 117566 Singapore, Singapore

Received 16 November 2007; received in revised form 30 January 2008; accepted 31 January 2008

Abstract

The coupling strategies for natural ventilation between building simulation (BS) and computational fluid dynamics (CFD) are discussed and coupling methodology for natural ventilation is highlighted. Two single-zone cases have been used to validate coupled simulations with full CFD simulations. The main discrepancy factors have also been analyzed. The comparison results suggest that for coupled simulations taking pressure from BS as inlet boundary conditions can provide more accurate results for indoor CFD simulation than taking velocity from BS as boundary conditions. The validation results indicate that coupled simulations can improve indoor thermal environment prediction for natural ventilation taking wind as the major force. With the aids of developed coupling program, coupled simulations between BS and CFD can effectively improve the speed and accuracy in predicting indoor thermal environment for natural ventilation studies.

Keywords: Coupled simulation; Building simulation; Computational fluid dynamics

1. Introduction

The methods for natural ventilation study to evaluate facade performance are categorized into three types: field measurements, controlled experiments and numerical simulations. Field measurements can only collect on site data from a few buildings, the locations of the instruments are restricted by on site conditions for the purpose of safety and security, and uncertainties of these measurements could be significant and thus make it difficult for further data analyses. Data obtained from a controlled environment such as wind tunnel experiments and full-scale model experiments are more reliable than those collected in field measurement. However, setting up and running these experiments are time consuming and high cost. The quality of the data acquired from these experiments is also limited by the accuracy of the instruments.

Numerical simulation is a cost-effective and efficient approach to predict thermal performances of facade in naturally ventilated buildings among various architecture designs. Simulation methods for natural ventilation fall into two broad categories: computational fluid dynamics (CFD) method and building simulation (BS) method. Building simulation can model heat transfer and radiation processes in seconds based on heat balance methods and CFD can predict reliable detailed airflow for outdoor and indoor.

CFD simulation provides detailed spatial distributions of air velocity, air pressure, temperature, contaminant concentration and turbulence by numerically solving the governing conservation equations of fluid flows. It is a reliable tool for the evaluation of thermal environment and contaminant distributions. These results can be directly or indirectly used to quantitatively analyze the indoor environment and determine facade system performances. However, the application of CFD for natural ventilation prediction has been limited due to long computational time and excessive computer resource requirements. Solar

*Corresponding author. Tel.: +65 65163423.
E-mail address: bdgwnh@nus.edu.sg (N.H. Wong).

0360-1323/$ - see front matter
doi:10.1016/j.buildenv.2008.01.015

radiation cannot be easily considered within CFD model and CFD simulations require high computation cost, especially when grid size requirements for various computational domains are inconsistent, such as, indoor and outdoor airflow simulation and heat conduction and airflow simulation.

BS tools basically include two fundamental modules: thermal simulation and airflow network to solve the heat and mass transfer and airflow in the building systems. These tools greatly facilitate energy-efficient sustainable building designs by providing rapid predictions of facade thermal behaviors, indoor airflow of the building and better understanding of the consequences of various design decisions. However, BS assumes the indoor air is well mixed. It can only provide the uniform results for targeted spaces, which normally does not meet the requirements for detailed indoor environment analyses. The simple flow-pressure governing equations for various components limit the accuracy of the airflow model. Information provided by these two programs (CFD and BS) is complementary for advanced evaluation of building designs for thermal comfort. The integration of BS and CFD programs can eliminate a few assumptions employed in the separate applications, dramatically reduce computation time of CFD, and result in accurate and quick predictions of building performance. Therefore, the integration of BS and CFD simulation is becoming an active research area in recent decades.

The previous research work for integration of BS and CFD can be mainly divided into two categories according to the coupling purposes: thermal environment predictions for air-conditioned rooms and thermal environment predictions for naturally ventilated rooms.

To accurately evaluate energy loads for air-conditioned room, several BS programs have been externally or internally coupled with CFD simulation program in the thermal aspect [1–5]. For internal coupling between BS and CFD, a module CFD was built into the BS and only one model is to be built for the coupled simulation. Matrix formation in each module (thermal analysis, airflow network or CFD) is affected due to data exchange in each time step. However, the internal coupling between BS and CFD strictly limited simulation geometry and the number of grids, and the parallel computing techniques used in current state-of-the-science CFD codes for speedy computation also cannot be employed. For external coupling between BS and CFD, there is no internal CFD module built inside BS. The changes of source code can be minimized. Simulation models should be built separately in CFD and ESP-r. Data exchanges for boundary conditions are needed to bridge the two programs.

Although, the integration methods for air-conditioned buildings to accurately estimate energy consumption in buildings are well studied by many researchers, there are limited investigations on the integration of CFD simulation and BS for naturally ventilated buildings for better thermal comfort.

Negro [1] built the internal CFD module into ESP-r and achieved the integration between internal CFD module and thermal simulation module to perform the detailed heating and cooling load calculations by data exchanges between thermal simulation and CFD. Internally CFD coupling with airflow network module in ESP-r was built. However, this internal coupling application is constrained by the requirement of cubic simulation domain, maximum grid sizes and computational time and convergence issues.

Carrilho da Graça et al. [6] used a coupled, transient simulation approach to model heat transfer and airflow in the apartments in Beijing and Shanghai. Wind-driven ventilation was simulated using CFD for each outside wind direction and velocity and the surface convection coefficients have been calculated by experimental correlations as a function of the air velocity near the walls suggested by Chandra and Kerestecioglu [7] to be used as boundary conditions for thermal analysis. Carrilho da Graça et al. used isothermal CFD calculations to avoid detailed calculation of airflow and indoor surface temperatures. As they stated that thermal buoyancy effects are much smaller then wind-driven pressure in naturally ventilated residential buildings, the coupling approach that airflow simulation is independent of thermal simulation by assuming uniform indoor air temperature is adopted. Occupant thermal comfort is accessed using Fanger's comfort model. The results show that night cooling may replace air-conditioning systems for a significant part of the cooling season in Beijing, but with a high condensation risk. But for Shanghai, neither night cooling nor daytime ventilation can be considered successful.

Sreshthaputra et al. [8] coupled DOE-2 program with transient HEATX (3D-CFD simulation program) for natural ventilation to analyze the heat transfer and airflow performance of an unconditioned 100-year-old Buddhist temple in an urban area of Bangkok, Thailand. Two variables were fed back and forth between two programs during the calibration process. On the one hand, the amount of outside air infiltration specified by the term "air change rate (ACH) in DOE-2 is specified according to CFD results. The CFD simulation was used to estimate the maximum ventilation rate to be supplied to DOE-2 by multiplying the maximum air velocity across the windows with the total area of the windows. On the other hand, the interior surface convection coefficients for each surface based on CFD results are transferred to DOE-2 as boundary conditions when the temperature difference for indoor air temperature between DOE-2 and CFD is larger than 1 °C. Since the whole year dynamic simulation is estimated to take very long time (730 days approximately) with the coupled simulation, the method where average values of the air exchange rates and the corresponding convection coefficients were obtained from the coupled simulation of 2 selected days was adopted. These average values were used by DOE-2 to perform the annual hourly calculations. However, although the computational