A convenient coupled simulation method for thermal environment prediction in naturally ventilated buildings

L. Wang, N. H. Wong
National University of Singapore, Singapore

ABSTRACT

As both CFD and building simulation have their own limitations in the thermal prediction of natural ventilation. External coupled simulation method between CFD (FLUENT) and building simulation (ESP-r) has been put forward for thermal environment prediction for naturally ventilated building design. The coupling mechanism and coupling program interface are introduced in the paper. Full CFD simulation and on site field measurements are done for the validation of the coupled simulation method. The results of coupled simulations show that the integration of CFD and building simulation can accurately and efficiently provide indoor thermal environment and can be used as a convenient tool for facade designs for naturally ventilated residential buildings. Key words: Coupled simulation, CFD, Building simulation

1. INTRODUCTION

The concept of natural ventilation is well accepted and welcomed by people and designers in the world, especially when the benefits of natural ventilation, including reducing operation cost, improving indoor air quality and providing satisfactory thermal comfort in certain climates, are being acknowledged. Even in hot-humid climates, where air-conditioners are common in both residential and commercial buildings, HDB (Housing & Development Board) residential buildings, where about 86% of people in Singapore live, are designed to be naturally ventilated. CFD simulation is one of the important methods for natural ventilation study. It can provide detailed air temperature, air velocity, contaminant concentration within the building or outdoor spaces. It has become a reliable tool for the evaluation of thermal environment and contaminant information. However, thermal comfort prediction of naturally ventilated buildings with CFD simulation only is not an easy task. Building simulation (BS) is another popular method for investigating naturally ventilated building design. Thermal simulation and airflow network are two fundamental modules in building simulation tool. With the aids of building simulations (BS), which provide rapid prediction of facade thermal behaviours through solving the heat and mass transfer and airflow network in the building systems in one dimension, the task for natural ventilation treatment can be simplified. However, the airflow network method for airflow estimation in building simulation cannot accurately predict indoor airflow. Therefore, how to effectively integrate the BS and CFD simulation to improve the accuracy to assess the performance of natural ventilation becomes a primary objective.

In this paper, coupling procedures and a coupling program interface are introduced and the developed coupling program is further validated with two methods: full CFD simulation (both indoor and outdoor airflow are calculated simultaneously) and field measurement results, which are presented as follows.

2. COUPLING PROCEDURES

Commercial software FLUENT is used as the external CFD program to couple with ESP-r (Clarke and McLean, 1988). A one-way coupling is adopted for wind-driven natural ventilation from ESP-r to FLUENT simulation. From building simulation, the inside surface temperatures for each wall, detailed boundary conditions for the openings including velocity boundary conditions (or pressure boundary conditions), airflow direction, air temperature are provided for detailed indoor CFD simulation. The predicted indoor average temperature by CFD simulation will be compared with zone temperature from building simulation results. A text-mode data exchange interface has been programmed to provide the convenient data passage between the two programs to automatically implement the data exchange between the two programs. Detailed thermal comfort prediction model for indoor environment is finally provided for evaluation in the data interface. Coupled simulations can be done automatically using the program interface. In the following section, coupled simulation results are to be validated with full CFD simulations and field measurement studies.

3. FULL CFD VALIDATION

In this full CFD validation, a multi-zone scenario was investigated for full CFD validation. Two cases (case 1: 2nd Jan 18:30 and case 2: 1st Jan 12:30), with wind direction 0°(N) and wind direction 45°(NE), selected from different weather conditions in Singapore weather file 2001 are simulated in the scenario and the climatic data
for both cases are shown in Table 1.

Table 1: Climatic data

<table>
<thead>
<tr>
<th>Case</th>
<th>Wind Direction (°N; 90°E)</th>
<th>Wind Speed (m/s)</th>
<th>Temperature (°C)</th>
<th>Relative Humidity (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>1.7</td>
<td>25</td>
<td>92</td>
</tr>
<tr>
<td>2</td>
<td>45</td>
<td>5.8</td>
<td>29</td>
<td>76</td>
</tr>
</tbody>
</table>

Single zone scenarios with coupling program have been investigated and validated with full CFD simulation (Wang and Wong, 2006). Multi-zone scenarios are more complicated than single zone scenarios since one of the opening in one zone as outlet (inlet) has been used to connect with another zone as inlet (outlet). Therefore, the boundary conditions for the internal opening are significant but difficult to predict. The comparison results of single zone scenarios show that taking pressure as opening boundary conditions is more accurate than taking velocity as opening boundary conditions (Wang and Wong, 2006). Therefore, for the multi-zone scenarios, only pressure boundary conditions are taken into consideration. There are three ways to treat internal openings, which connect two nodes (rooms). The first one is called the “pre” method. In this method, indoor CFD simulation is carried out in one of the two rooms that we are interested in. The internal opening boundary condition is taken from the neighbouring node pressure (the pressure of the other room) as internal opening boundary condition in the “pre” method. The second method is called the “pre-ave” method. Similar to the “pre” method, indoor CFD simulation is carried out in one particular room, but the pressure boundary conditions for internal openings are taken as the average value of two connected nodes. The third method is called “pre-room” method. In this method, the whole connected zones are in the computation domain for indoor CFD. Therefore, the internal openings are treated in CFD as default openings and no pressure values need to be assigned to them beforehand. The inaccuracy originated from the estimation of internal opening pressure could be eliminated using the third method, the whole connected zones are in the computation domain. Therefore, for the multi-zone scenarios, only pressure values need to be assigned to them beforehand. The comparison results of single zone scenarios for indoor CFD simulation. The area weighted velocity magnitude results at the height of 1.5 m shown in Table 2 are compared among full CFD simulation, coupled ESP-r and indoor CFD simulation in a particular zone (pre, pre-ave, pre-room), ESP-r simulation only. The area-weighted air velocity for full CFD simulation is the indoor averaged air velocity at a certain level, and for ESP-r is defined by Equation 1

\[ V = \frac{\dot{m}}{\rho A} \]  

(1)

where \( V \) (m/s) represents the area-weighted indoor air velocity, which is unique in multi-zone airflow simulation with well-mixed assumption, \( \dot{m} \) (kg/s) is the total mass flow rate, \( \rho \) (kg/m³) is the air density from the zone where the air flows into, and A (m²) is the section area perpendicular to the airflow.

The ESP-r simulation results provide boundary conditions for indoor CFD simulation. The area weighted velocity magnitude results at the height of 1.5 m shown in Table 2 are compared among full CFD simulation, coupled ESP-r and indoor CFD simulation in a particular zone (pre, pre-ave, pre-room), ESP-r simulation only. The area-weighted air velocity for full CFD simulation is the indoor averaged air velocity at a certain level, and for ESP-r is defined by Equation 1.
along height and length for living zone. It can be seen from comparison results that with proper coupling between the building simulation and CFD program with pre-ave and pre-room methods, the indoor airflow profile can be well predicted for natural ventilation. The results show that the coupling program results are consistent with the full CFD simulation results. The coupling program can better predict the average velocity in the room than using building simulation program alone. However, coupled simulations may not be able to accurately predict the velocity profile near the inlet. In addition, the pre method shows the largest errors between full CFD and coupled simulations.

Table 2: Result comparison

<table>
<thead>
<tr>
<th>Area Weighted velocity (1.5m Height)</th>
<th>Full CFD simulation (m/s)</th>
<th>Indoor CFD with pressure inlet (m/s)</th>
<th>Indoor CFD with pressure -ave (m/s)</th>
<th>Indoor CFD with pressure -room (m/s)</th>
<th>ESP-r (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1: 2nd Jan 18:30</td>
<td>0.17048</td>
<td>0.2633</td>
<td>0.2214</td>
<td>0.1954</td>
<td>0.091</td>
</tr>
<tr>
<td>Case 2: 1st Jan 12:30</td>
<td>0.56018</td>
<td>0.6999</td>
<td>0.5740</td>
<td>0.64039</td>
<td>0.286</td>
</tr>
</tbody>
</table>

Figure 2 Area_weighted velocity results comparison along vertical (z) direction and length (y) direction for living room in case 1 among full CFD simulation, indoor CFD simulation with average pressure boundary condition, indoor CFD simulation with pressure boundary condition, and indoor CFD simulation for the whole room.

Figure 3 Area_weighted velocity results comparison along vertical (z) direction and length (y) direction for living room in case 2 among full CFD simulation, indoor CFD simulation with average pressure boundary condition, indoor CFD simulation with pressure boundary condition, and indoor CFD simulation for the whole room.

4. VALIDATION WITH FIELD MEASUREMENT

The HDB block601 is located in Clementi, West coast. It is facing 18 degree northwest and the height of the building is 38.7 meters. A sixth level 4-room unit has been used for this field measurement. The digital multi-channel data logger of BABUC has been used to record indoor thermal parameters, including dry bulb temperature, indoor velocity, relative humidity, globe temperature and wet bulb temperature. The measurement was taken at the centre point of the living room at 20 minute interval. Eight thermal coupling wires (T type) have been used to measure the surface temperature both inside and outside of external wall in living room and kitchen room at 10 minute interval.

There are three bedrooms in the unit (Figure 4). For the purpose of validation, the doors of bedroom connected with the living room are closed. The internal doors are in good airtightness. Therefore, the airflow network in the unit comprises two boundary nodes (north and south nodes) and two internal nodes (living room node and kitchen node), as illustrated in Figure 4. The airflow route of the unit through the living room and kitchen room is North->living room -kitchen ->South or South->kitchen->living room->North.
The weather data were obtained from NUS weather station, located on the rooftop of building E2 (Faculty of Engineering) of the National University of Singapore, Kent Ridge campus. The geographical coordinates are approximately: 1 deg 18 min N (latitude) and 103 deg 46 min E (longitude). The vertical height of the weather station above the ground is 25.3 m (at the cup anemometer position). The simplified equation (2) is used to describe the relationship between building height wind speed and wind speed measured from Meteorological station at a standard height of 10m.

\[ \frac{v_z}{v_{10}} = k \frac{z_l}{a} \]

\[ v_z \] ---building height wind speed (m/s)

\[ v_{10} \] ---wind speed measured from Meteorological station at a standard height of 10m (m/s)

\[ z_l \] ---building height (m)

\[ k, a \] ----constants dependent on terrain. Here, we use the terrain coefficients for urban, \( k=0.35 \) and \( a=0.25 \).

Because the NUS weather station is located on a small ridge, however, the positioning of the instruments does not conform to WMO guidelines (WMO, 1983) for weather stations in open terrain or recommendations for urban areas. We use the modified wind speed equation (3) for the building height wind speed.

\[ \frac{v_z}{v_{nm}} = \left( \frac{z_l}{z_{nm}} \right)^a \]

\[ v_{nm} \] --- wind speed measured from NUS weather station (m/s)

\[ z_{nm} \] ---NUS weather station height (25.3 m)

4.1 ESP-r simulation

A HDB building has been modelled in ESP-r (a 14-storey residential building). The subject of the ESP-r simulation is the living room of a HDB unit on the sixth floor. The purpose of the simulation is to compare the simulation results with field measurements. There are altogether 22 zones in the building model. Each neighbouring level of the sixth level, such as the fourth level, fifth level, seventh level and eighth level, are divided into two zones, including left zone and right zone. The sixth level includes 10 zones for each specific unit layout. The other distant levels have to be combined together due to the limitation of maximum-zones and geometric model in ESP-r. The windows in living room and in kitchen area are adopted half area (1.44m²) as airflow network components. The building material setting is consistent with the specific materials in the real building. The solar absorbance of external walls is set to 0.3 (white color surface) and the emissivity is set to 0.9. The simulation results of ESP-r compared with field measurement have been shown in Figures 5-9. The comparison results indicate that the internal surface temperature and external surface temperature calculated with ESP-r have been compared with those measured surface temperature (shown in Figures 5 and 6). The comparison results indicate that the internal surface temperature with building simulation agrees well with the measuring data and the external surface temperature shows fluctuations around the measured external surface temperature. The relative humidity simulation results are well fitted with measuring results (Figure 7). The comparison results for dry bulb temperature for the living room are shown in Figure 8. From the results, it can be seen that building simulation tends to predict slightly higher zone temperature than field measurement results. Figure 9 shows the indoor air velocity results between field measurement and building simulation. As we can see from the results, the predicted indoor air flow is much lower than the measuring data.

Figure 4: The layout of the four-room HDB unit

Figure 5: Internal surface temperature of living room comparison between measurement and building simulation

Figure 6: External surface temperature of living room comparison between measurement and building simulation
The predicted area-weighted velocity at 1.5m above the floor and the predicted indoor air velocity at the specific measuring point are obtained by using BS-CFD coupling program for a time series simulation. Compared with the field measurement results, the predicted indoor air velocities by ESP-r are quite low, although the results show good correlation between the predicted velocity by ESP-r and measured data. These results confirm that building simulation (airflow network) cannot accurately predict mass airflow rate and indoor air velocity. This can be attributed to simply using a set of empirical equations governed by pressure differences for mass flow rate prediction of different kinds of components connecting airflow in the building simulation. Therefore, mass flow rate passing by each component cannot be accurately estimated. The underestimation becomes more obvious especially for cross ventilation conditions, under which air velocities around openings and indoor air flow rates can be highly increased. In the airflow network simulation, mass flow rates are calculated based on empirical equations related with the types of components and only mass conservation equation is considered, but momentum equation and turbulence equation are not reflected in the airflow network. It can be seen from the comparison results (Figure 10) that with the use of coupling program, the indoor air velocity prediction can be largely improved and the predicted air velocity with BS-CFD coupling program can provide the designers more reliable indoor air velocity results. Figure 11 shows the comparison results of indoor air dry bulb temperature among field measurement results and ESP-r simulation results and coupling program results from Dec 26 – Dec 29, 2005. The area-weight indoor air dry bulb temperature and the dry bulb temperature at the specific measuring point are obtained by using BS-CFD coupling program. From the figure, it can be seen that the predicted dry bulb temperature by ESP-r is quite close to that predicted by coupling program, which are mainly controlled by outdoor ambient temperature. From the comparison results, it can be concluded that quasi-steady coupling program could largely improve the accuracy of indoor air velocity prediction and further improve thermal comfort evaluation.
5. CONCLUSION

A coupling program between ESP-r and FLUENT and an automatically running text-mode interface was developed. The validation results by full CFD simulation and field measurements indicate that the coupling program can improve indoor thermal environment prediction for better prediction of natural ventilation taking wind as the major force. Comparing with full CFD simulation (both indoor and outdoor airflow simulation), the coupling program can largely reduce the computational cost and the impacts of solar radiation on facade heat gains could be easily considered. On the other hand, it can improve the accuracy airflow prediction for wind driven natural ventilation based on building simulation program. The coupling program between BS and CFD simulation can improve the accuracy and efficiency for predicting indoor thermal environment in detail and can be used as a convenient tool for facade design evaluation of naturally ventilated buildings.

REFERENCES

FLUENT user’s guide version 6.2. FLUENT Inc.