Natural ventilation simulation with coupling program between building simulation (BS) and computational fluid dynamics (CFD) simulation program for accurate prediction of indoor thermal environment

Wang Liping and Wong Nyuk Hien

Department of Building, National University of Singapore, Singapore, 117566

ABSTRACT: There is no available tool to better predict natural ventilation design. In order to accurately and quickly assess the natural ventilation, the coupling program between building simulation (BS) and computational fluid dynamics (CFD) simulation is developed. Coupling program was validated with two different methods: full CFD simulation and field measurement. In the full CFD validation, a multi-zone case was investigated to validate coupling simulation results with full CFD results. In the field measurement validation, the measuring results for a typical naturally ventilated HDB building in Singapore has been used to compare with coupling simulation results. It was concluded that the building simulation alone is difficult to accurately predict indoor thermal environment. This paper suggest that coupling program between BS and CFD is able to accurately and quickly predict natural ventilation by taking pressure boundary conditions for indoor CFD simulation.

Keywords: Coupling stages, CFD program, Building simulation, Natural ventilation

1. INTRODUCTION

Passive cooling of houses using natural ventilation has become an attractive alternative to alleviate problems associated with air-conditioning such as energy shortage, global warming and sick building syndromes. Various mechanical systems including heating, ventilation and air-conditioning (HVAC) systems in residential and office buildings contribute substantially to the energy consumption. In the United States, building services use more than one third of the primary energy consumption and two thirds of all electricity (U.S. Energy Information Administration 1995).

As the benefits of natural ventilation, including reducing operation cost, improving indoor air quality and providing satisfactory thermal comfort in certain climates, are being acknowledged, there is a growing interest in the natural ventilation of buildings. The concept of natural ventilation is well accepted and welcomed by people and designers in the world. Even in hot-humid climates, where air-conditioners are common in both residential and commercial buildings, HDB (Housing Developing Board) residential buildings, where about 86% people in Singapore live, are designed to be naturally ventilated.

However, natural ventilation is difficult to design and control although the principle itself is simple. Furthermore, because of the excessive amount of moisture in the air and intensive solar radiation, many passive cooling design strategies are difficult to be realized in hot and humid regions. The success of a naturally ventilated building is decided by a good indoor climate, which influences its sustainability. For this reason, naturally ventilated building design in hot-humid climates need to pay more attention to orientations, shading devices, material selections, window sizes and positions. Therefore, the optimization and evaluation of the facade system in naturally ventilated buildings are significant in natural ventilation designs. Because of time and budget constraints, only few cases could be measured on site or modelled in wind tunnel experiment. Large amount of parametric study on facade can only depend on simulations. However, there are no available tools which could accurately predict natural ventilation for various facade designs.

Currently, simulation methods for natural ventilation could be divided into two types: Computational fluid dynamics method (CFD) and building simulation method (BS). CFD simulation could 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, the application of CFD for natural ventilation prediction has been limited due to long computational time and excessive computer resource requirements. On the other hand, building simulation (BS) tools could greatly facilitate energy efficient sustainable building design by providing rapid prediction of facade thermal behaviours, indoor air flow of the building and better understanding of the consequences of various design decisions, through solving the heat and mass transfer and airflow network in the building systems. However, BS programs assume the indoor air is completely mixed and uniform; so BS results could only provide the uniform results for targeted spaces, which normally does not meet our requirements for detailed indoor environment evaluation. The airflow network method for airflow estimation in building simulation can not accurately predict indoor airflow. Therefore, the integration of the BS and CFD simulation could provide a quick and more accurate way to assess the performance of natural ventilation in whole buildings, as well as detailed thermal environmental information in some particular spaces. There is urgent need to provide an efficient coupling program between BS and CFD to predict natural ventilation efficiently and accurately.

2. COUPLING STRATEGIES

As both indoor CFD simulation and building simulation have their own disadvantages, the integration of building simulation and CFD simulation is becoming an active research area in recent decades. CFD program has been integrated into building simulation for air conditioned rooms to improve the evaluation of building energy consumption (Negrao, 1995; Zhai *et al*, 2002). Djunaedy (2005) further extended the coupling program to external coupling between ESP-r thermal simulation with commercial software FLUENT for mechanical ventilation. For natural ventilation, Tan and Leon (2005) coupled a multi-zone airflow simulation program (Multi-Vent) with CFD (PHOENICS) by static strategy. However, the coupling program cannot well predict wind-driven natural ventilation alone. Good accuracy of the integration could only be obtained when buoyancy effects are involved in natural ventilation. On the other hand, for current high-rise residential buildings, the main driving force of natural ventilation is the wind. Nevertheless, currently, there is no coupling program to accurately and efficiently predict wind-driven natural ventilation.

A coupling program between building simulation program and CFD simulation is developed. Figure 1 shows the detailed coupling method between building simulation and indoor CFD program. The function of building simulation undertakes the thermal calculation of the effects of heat conduction of walls and solar radiation on the room and the function of indoor CFD simulation undertakes the indoor convective heat transfer and detailed indoor airflow simulation. In terms of data exchange, from building simulation, inside surface temperatures for each wall will be fed into indoor CFD simulation and detailed boundary conditions for the openings including velocity boundary conditions (or pressure boundary conditions), airflow direction, outdoor air temperature will be provided for detailed indoor environment calculations. The predicted indoor average temperature by CFD simulation will be compared with zone temperature from building simulation results. The data exchange interface has been made to provide the convenient passage between the two programs, which can keep the code changing to minimum of building simulation—Esp-r. Commercial software FLUENT has been used to be the exterior CFD program. The data exchange Interface has been programmed, which can automatically realize the data exchange and comparison between them. The detailed thermal comfort index for indoor environment is finally provided for evaluation in the data interface.

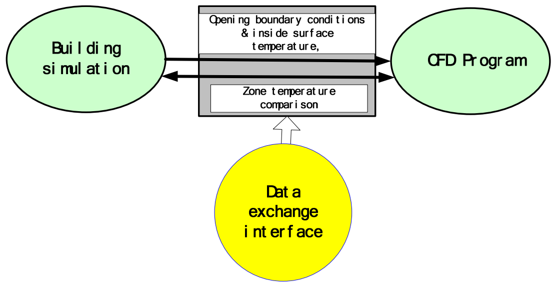

Figure 1: the coupling strategy between building simulation and CFD

One of the important tasks for natural ventilation simulation is to accurately predict the indoor airflow (airflow contour, vector, and average velocity) which cannot be done by multi-zone simulation. Therefore, the boundary conditions for indoor CFD simulation, obtained from building simulation results, are significant to determine the indoor airflow profile. There are two types of inlet conditions could be defined in CFD simulation for incompressible indoor airflow. One is pressure inlet condition, the other one is velocity inlet condition. The total pressure condition or velocity condition at openings are derived from the building simulation results.

The total pressure conditions for wind induced boundary nodes are fixed and calculated by using pressure coefficient in building simulation. The total pressure conditions in ESP-r for boundary nodes are defined as (ESP-r, 2001)

$$P = Cp \times \frac{1}{2}\rho WS^2 - 9.81 \times \rho \times H$$

$$\rho = 1.1881 \times 293.15/(273.15 + T)$$

Where P indicates the total pressure of the boundary nodes (which includes stack pressure), ρ indicates the outdoor air density by using Bussinesq approximation, which is accurate as long as actual density changes are small, WS indicates the wind speed at the reference height (roof height), and H indicates the height of the boundary nodes.

The internal opening as a component between two zones adopts the calculated pressure value as the opening pressure value. For indoor CFD simulation, by comparing pressure values, the opening with larger pressure value will be taken as pressure-inlet, the other opening with smaller value will be taken as pressure-outlet condition. If the internal opening is taken as inlet, the inlet direction is normal to the boundary. Otherwise, the wind direction will be taken as inlet direction for external openings.

The velocity condition at the opening can be calculated directly from ESP-r calculation results, which is defined as

$$V_{i,j} = \frac{\dot{m}_{i,j}}{\rho \cdot A}$$

Where $V_{i,j}$ represents the component velocity from ith node to jth node, $\dot{m}_{i,j}$ indicates the total mass flow rate between ith node and jth node, ρ indicates ith zone air density if $\dot{m}_{i,j}$ >0, else, ρ indicates jth zone air density if $\dot{m}_{i,j}$ <0, A indicates the component area. For indoor CFD simulation, by comparing the signal of velocity values, the one with positive signal for ith zone is taken as velocity inlet, the other one is taken as outflow. Similar to pressure boundary conditions, if the internal opening is taken as velocity inlet, the inlet direction is normal to the boundary. Otherwise, the wind direction will be taken as inlet direction for external openings.

The primarily study on coupling program shows that taking the pressure as the opening boundary conditions for coupling program between BS and CFD can provide more reliable prediction results than taking velocity as boundary conditions(Wang and Wong, 2006).

The developed coupling program is validated with two methods: full CFD simulation and field measurement results, which are presented as follows.

3. FULL CFD VALIDATION

In this full CFD validation, a multi-zone scenario was investigated for full CFD validation. The multi-zone case represents a typical HDB unit in Singapore, which was simulated by using full CFD simulation and coupling program between BS and CFD. Two cases (case 1: 2nd Jan 18:30 and case 2: 1st Jan 12:30) selected from different weather conditions in Singapore weather file 2001 are investigated and the climatic data for both cases are shown in Table 1.

Table 1: Climatic data

Time	Wind Direction (°)	Wind Speed (m/s)	Temperature (□)	Relative Humidity (%)
Case 1	0	1.7	25	92
Case 2	45	5.8	29	76

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 in this internal opening are significant and difficult to predict. Since 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). Here for multi-zone scenarios, only pressure boundary conditions are taken into consideration. Two types of pressure boundary conditions have been tested according to the way to treat internal openings. One way for the multi-zone scenario is to take the average pressure value of two neighbouring zones, named by pre-ave. Another way for multi-zone scenario simulation with coupling program is to take the whole connected zones into indoor CFD simulation, named by pre-room, and therefore the internal openings will be treated by CFD as an internal hole. The inaccuracy originated from the estimation of internal opening pressure could be minimized for the second strategy. Therefore, a multi-zone scenario--a typical residential flat in Singapore--has been investigated in these types of boundary conditions and validated with full CFD simulation.

The layout of the typical flat is shown in Figure 2. Totally eight nodes have been used for airflow network simulation in ESP-r as illustrated in Figure 2. They are living room, common1, master room, kitchen, and another four boundary nodes (north, south_common1, south_master room and south_kitchen).

The coupling program between ESP-r building simulations and indoor CFD simulation are processed with multi-zones or a particular zone. The full CFD computations for both indoor and outdoor airflow simulation have been used for validation. The velocity inlet boundary condition for both indoor and outdoor CFD simulation is using the atmosphere boundary profile for inlet boundary condition. Pressure-outlet boundary condition has been adopted for the full CFD simulation domain.

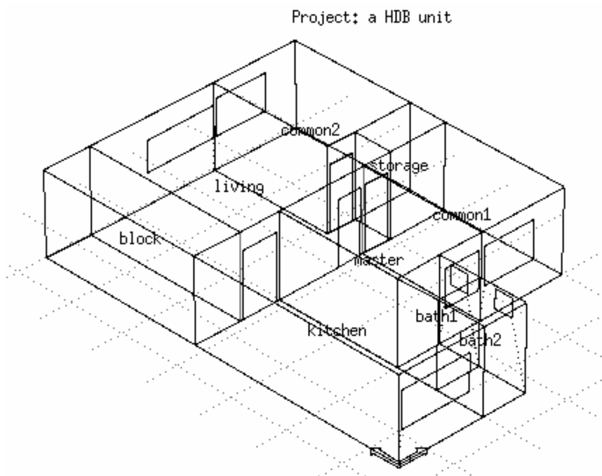

Figure 2 A HDB flat in Singapore layout

The ESP-r simulation results are used for indoor CFD simulation. The area weighted velocity magnitude results at the height of 1.35 m are illustrated in Table 2 for comparisons among full CFD simulation, ESP-r with indoor CFD simulation in a particular zone (pressure inlet), ESP-r with indoor CFD simulation in a particular zone (pressure-average inlet), ESP-r with indoor CFD simulation in multi-zones and ESP-r simulation.

Table 2: Results comparison (living room)

Area Weighted room velocity (m/s)	Full CFD simulation	Indoor CFD with pre-ave	Indoor CFD with pressure-room	ESP-r
Case 1	0.2193	0.31125	0.2652	0.0539
Case 2	0.75	0.9894	0.6747	0.1698

The comparison results in Table 2 indicate that building simulation alone could not well predict indoor air flow rates, the indoor airflow prediction could be improved for natural ventilation with coupling between building simulation program and CFD program. The detailed area weighted velocity magnitude results are compared in Figure 3 along height of the room and length of the room for case1 and Figure 4 for case 2. From the results, it can be seen that the coupling program by adopting pre-room strategy could better predict the average velocity in the room than coupling program by adopting pre-average strategy. However, the coupling program may not be able to accurately predict the velocity profile near the inlet, especially when jet flow is not perpendicular to the openings. The major differences among three different scenarios (full CFD simulation, ESP-r with velocity inlet CFD simulation, and ESP-r with pressure inlet CFD simulation) appear at the inlet boundary conditions.

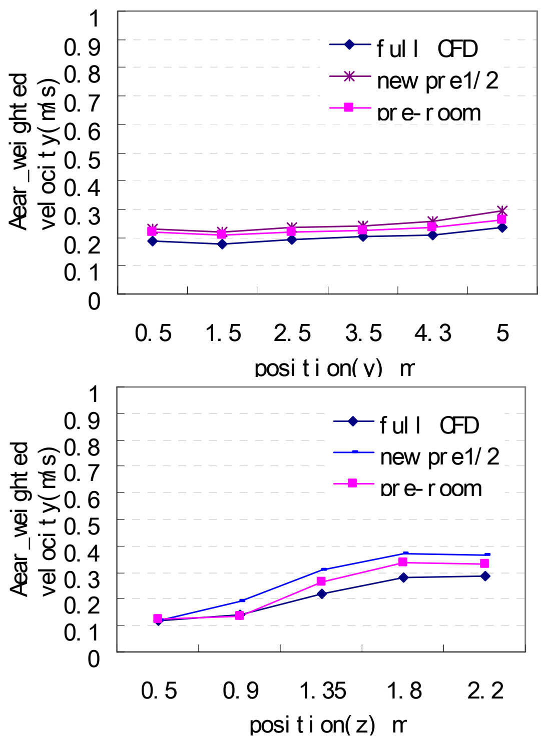

Figure 3: Area_weighted velocity results comparison along vertical (z) direction and length (y) direction for living room in case 1 among full CFD simulation, indoor CFD simulation with average pressure boundary condition and indoor CFD simulation for the whole room.

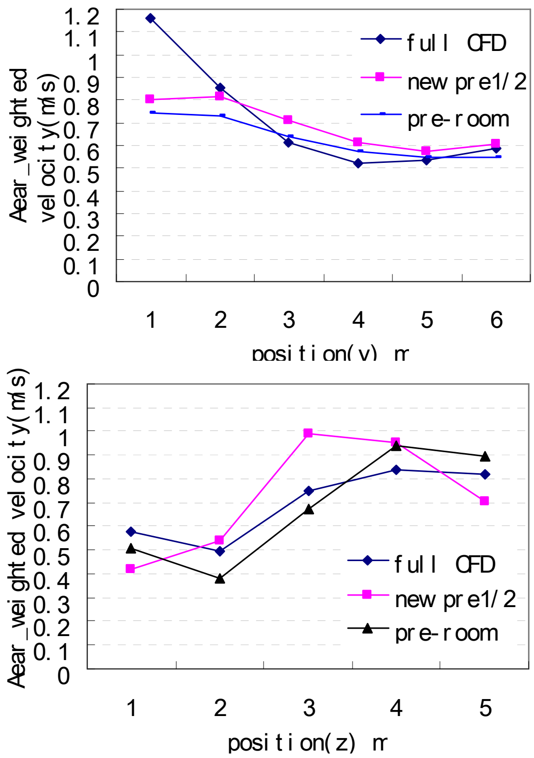

Figure 4: Area_weighted velocity results comparison along vertical (z) direction and length (y) direction for living room in case 2 among full CFD simulation, indoor CFD simulation with average pressure boundary condition and indoor CFD simulation for the whole room.

4. VALIDATION WITH FIELD MEASUREMENT

4.1 Cp prediction with external CFD simulation

The purpose of this study is to predict the pressure coefficient for high-rise residential buildings. Pressure coefficient (Cp) of external surfaces is a significant parameter for accurate prediction of natural ventilation with coupling program between BS and CFD.

External airflow simulations were carried out for a typical HDB building- Block 601 in Singapore to obtain the Cp values at the sixth floor. Eight cases have been simulated for predicting pressure coefficient, with each has a incidence angle of 0°, 45 °, 90 °, 135 °, 180°, 225 °, 270 °, 315° from the building (Figure 5). A pressure coefficient set in building simulation typically comprises 16 values at 22.5 ° intervals. Therefore, the other eight pressure coefficients are interpolated from the obtained values by CFD simulation. The dimensions of the building cluster are around 90.4m (L) x 13m (W) x 38.7m (H). Therefore, the computational domain for windward part is around 500m, for each side part is around 400m, and for leeward part is around 700m. This makes sure that the turbulence flows behind the building are fully developed. A total number of two million mesh elements are generated for each case and the meshes around the building is illustrated in Figure 6.

The pressure coefficients of external surface temperature could be calculated by the following equation:

$$C_p = \left(\frac{p - p_\infty}{1/2 \rho V_r^2} \right)$$

Where p is the pressure, p_∞ is the pressure at free stream and V_r is the reference wind speed above the roof. The predicted Cp results will be input into the building simulation, taken as boundary conditions.

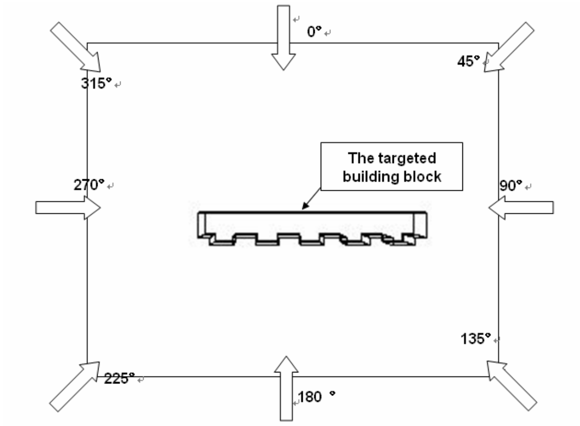

Figure 5: Computational methodology for various wind directions

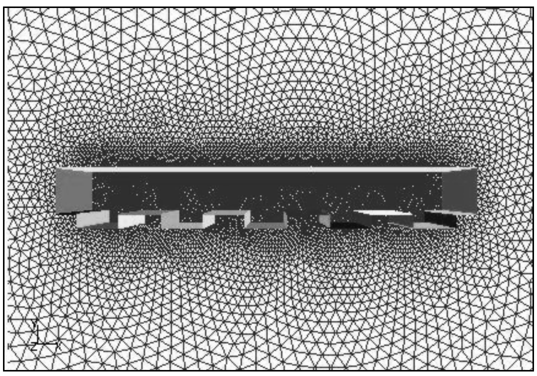

Figure 6: building model in west coast built in GAMBIT

4.2 field measurement

The HDB block601 is located in Clementi, West coast, as shown in Figure 7(a). 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 center point of the living room as shown in Figure 7 (b) 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, as shown in Figure 7 (c).

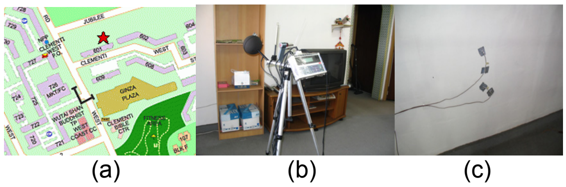

Figure 7: (a) Block 601 and surrounding buildings (b) Babuc layout in the living room (c) Thermal couple wires for surface temperature

There are three bedrooms in the unit (Figure 8). For the purpose of validation, the doors of bedroom connected with the living room are closed. Namely, 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 construction material properties are listed in Table 3.

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 deg18 min N (latitude) and 103 deg 46 min E (longitude).

Table 3: Facade material properties

	Material	Thickness (mm)	Thermal transmittance (W/km^2)	Name in ESP-r
North /South external wall	Concrete hollow block wall	100	1.89	RCwall
Internal wall	Concrete hollow block wall	100	1.89	RCwall
Gable End wall	Gable end cavity wall	260	1.29	Gable wall
Glass	6mm antisun glass	6	5.44	Single glass

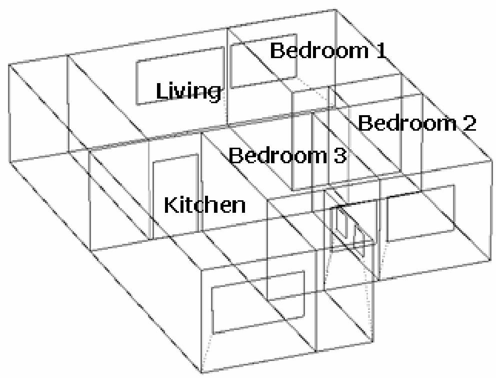

Figure 8: the layout of the four-room HDB unit

4.3 ESP-r simulation

ESP-r (ESP-r, 2001) is an integrated building simulation program including thermal, visual, acoustic, and energy assessment. 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 (Figure 9). 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 zone are adopted half area ($1.44m^2$) as airflow network components. The building material setting is consistent with the specific materials in the real building. The solar absorptance of external walls is set to 0.3 (white color surface) and the emissivity is set to 0.9.

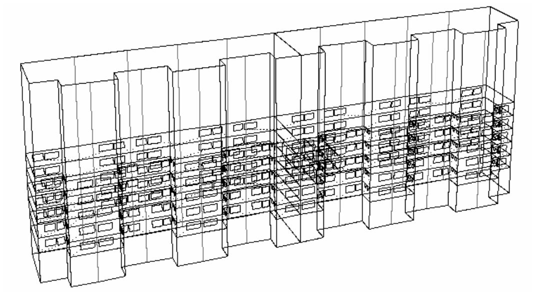

Figure 9: HDB block601 ESP-r model

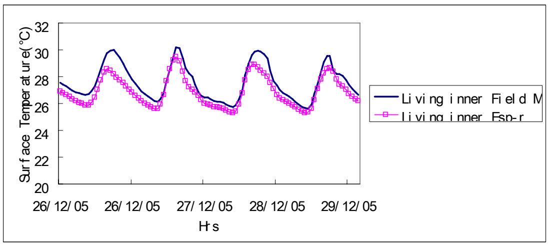

Figure 10: Internal surface temperature of living room comparison between measurement and building simulation

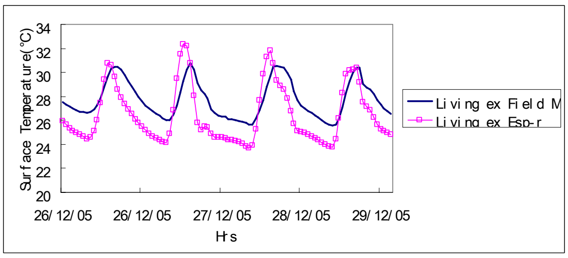

Figure 11: External surface temperature of living room comparison between measurement and building simulation

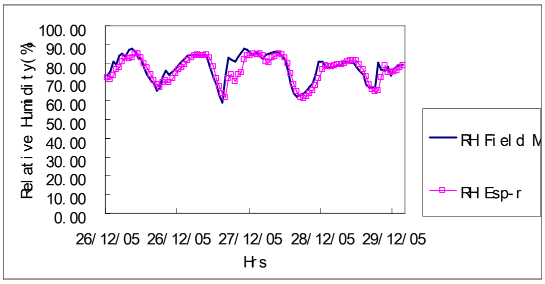

Figure 12: Relative Humidity result comparison between measurement and building simulation

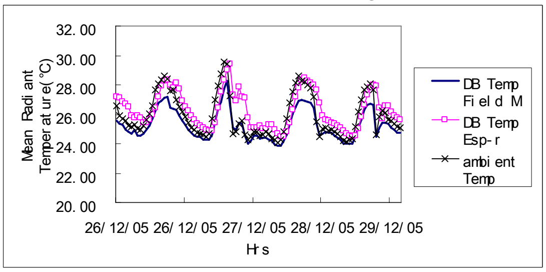

Figure 13: Dry bulb temperature result comparison between measurement and building simulation

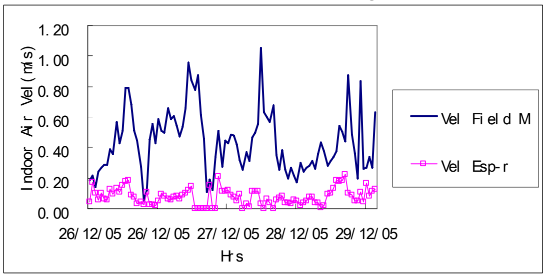

Figure 14: Indoor air velocity result comparison between measurement and building simulation

The simulation results of ESP-r compared with field measurement have been shown in Figure 10 - 14. The internal surface temperature and external surface temperature calculated with ESP-r have been compared with those measured surface temperature (shown in Figure 10 and Figure 11). 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 12). The comparison results for dry bulb temperature for the living room are shown in Figure 13. From the results, it can be seen that building simulation tends to predict slightly higher zone temperature than field measurement results. Figure 14 shows the indoor air velocity results between field measurement and building simulation. The indoor velocity has been estimated from the inlet velocity calculated by ESP-r using the following equation. As we can see from the results, the predicted indoor air flow is much lower than the measuring data.

$$V(m/s) = \frac{Inlet\ Velocity \times Inlet\ Area}{\sec tion\ area\ of\ the\ room} \quad (m/s)$$

4.4 Coupling simulation results

It has been noticed that the multi-zone model in ESP-r could not accurately predict the indoor air velocity, which would probably affect the prediction of other indoor thermal parameters. Therefore, to improve the accuracy of natural ventilation prediction, the coupling program between ESP-r and CFD are adopted. The predicted indoor environment (indoor air velocity and dry bulb temperature) with coupling program between building simulation (Esp-r) and computational fluid dynamics(CFD), has been compared with the field measuring data as shown in Figure 15 and Figure 16.

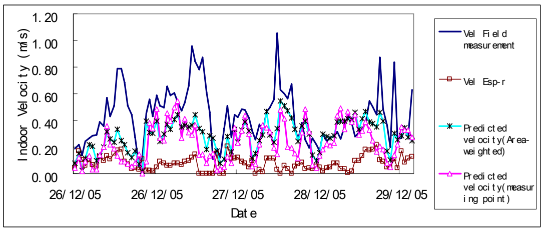

Figure 15: Indoor air velocity comparison results among Field measurement, Esp-r simulation (building simulation only) and Esp-r-CFD simulation (coupling simulation)

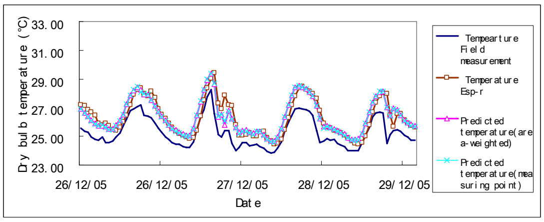

Figure 16: Indoor air temperature results among Field measurement, Esp-r simulation (building simulation only) and Esp-r-CFD simulation (coupling simulation)

Figure 15 shows the comparison results of indoor air velocity among field measured data and Esp-r simulation results and coupling program results from Dec 26, 2005 to Dec 29, 2005. The field measurement results are taken at the level of 1.5m above the floor near the centre of the room. 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 building simulation Esp-r are quite low, although the results show good correlation between the predicted velocity by Esp-r and measured data. These results appear to confirm that building simulation (airflow network) could not accurately predict mass airflow rate and indoor air velocity. This could 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 may not be accurately estimated, and this underestimation becomes more obvious especially for cross ventilation conditions, under which air velocities around openings and indoor air flow rates could be highly increased. In the airflow network simulation, mass flow rate are calculated based on empirical equations related with the types of component 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 15) that with the use of coupling program, the indoor air velocity prediction could be largely improved and the predicted air velocity with BS-CFD coupling program could provide the designers more reliable indoor air velocity results. However, at the peak measuring values in the daytime, discrepancies between field measurement and coupling program predicted velocity are observed. Generally speaking, it is a characteristic of the wind that greater fluctuations in the speed of gusts occur when the mean wind speed is high than when it is low. Normally, in the field measurement, the peak values indicate very high fluctuations of indoor air velocity at those periods. However, the coupling program takes uniform steady pressure boundary conditions for apertures at each time interval, which is applicable for most of the indoor time, but for highly fluctuating cases, the pressure conditions at the openings could not be simply taken at a uniform constant value in each time interval. Therefore, for the accurate prediction of highly fluctuating cases, unsteady full CFD simulations for both indoor and outdoor simulation at small time interval or large eddy simulation may be required, but computational time will be largely increased. In general, it could be concluded that BS-CFD coupling program could predict indoor air velocity in a more accurate way.

Figure 16 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

The coupling program could 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 could largely reduce the computational cost and the impacts of solar radiation on facade heat gains could be easily considered. On the other hand, it could improve the accuracy airflow prediction for wind driven natural ventilation based on building simulation program. It could be an optimum way to obtain the indoor thermal environment for naturally ventilated buildings quickly and accurately.

REFERENCES

Djunaedy (2005) External coupling between building energy simulation and computational fluid dynamics. Ph.D. thesis. Technische University of Eindhoven, April, 2005

Feriadi,H., Wong, N.H., Sekhar, C, and Cheong, K.W. (2003). Adaptive behaviour and thermal comfort in Singapore's naturally-ventilated housing Building Research and Information Journal. Vol. 31(1) 13-23.

Negrao (1995) Conflation of Computational Fluid Dynamics and Building Thermal Simulation. PhD Thesis. University of Strathclyde.

Tan,G. and Leon,G. (2005). Gang Tan and Application of integrating multi-zone model with CFD simulation to natural ventilation prediction Energy and Buildings.Vol37.(10)1049-1057.

Wang, L. and Wong N.H. (2006). Coupling between CFD and building simulation for better prediction of natural ventilation. 2nd INTA conference, Jogjakarta, Indonesia, 3-5 April,2006.

Zhai, Z., Chen, Q., Haves, P., and Klems, J (2002). On approaches to couple energy simulation and computational fluid dynamics programs Building and Environment. Vol 37(8-9) 857-864.