INNOVATIONS IN VENTILATION TECHNOLOGY

21ST ANNUAL AIVC CONFERENCE
THE HAGUE, NETHERLANDS, 26-29 SEPTEMBER 2000

INTEGRATION OF INDOOR AND OUTDOOR AIRFLOW STUDY FOR NATURAL VENTILATION DESIGN USING CFD

Z. Zhai, S.D. Hamilton, J. Huang, C. Allocca, N. Kobayashi, and Q. Chen*

Building Technology Program
Massachusetts Institute of Technology
77 Massachusetts Avenue, Cambridge, MA 02139-4307, USA

*Corresponding author. Phone: (617)253-7714, Fax: (617)253-6152, Email: qchen@mit.edu

ABSTRACT

Natural ventilation is one of the most fundamental techniques to reduce energy usage in buildings. However, due to complicated site plans and building layouts, it is difficult to design optimal layouts for the enhancement of ventilation without knowledge about the flow patterns. The employment of computational fluid dynamic (CFD) tools in the design process can give predictive feedback to the designers, allowing them to optimize airflow around the site to decide on building placement, orientation, and interior space layout. Since the simultaneous simulation of the building and the site plan would require large computer resources, this paper attempts to reduce the computing burden by decoupling outdoor flow modeling from an indoor airflow modeling, through an iterative design procedure. The decoupling uses loose and compact integration methods. The former method involves an outdoor simulation, from which indoor boundary conditions are extracted, while the latter simulates indoor and partial outdoor flow. An apartment complex in Shanghai is used as the model in this study.

Keywords: natural ventilation, CFD, airflow around buildings, indoor environment

INTRODUCTION

Natural ventilation is an essential part of sustainable building design. Energy-conscious designers harness the cooling capacity of natural winds to increase indoor thermal comfort and ultimately lessen the necessity for active space conditioning. By numerically solving a series of conservation equations related to mass, momentum, and energy, CFD tools help designers predict detailed airflows for special design cases and therefore plan a building with optimal natural ventilation. A number of papers show the effectiveness of this type of effort [1-3]. The CFD tools have clearly demonstrated to be superior to conventional wind tunnel tests due to their low costs and high efficiency. Although the CFD tools have some problems, such as the uncertainties in turbulence modeling, the technology has become more and more popular. Hence, this paper will discuss how to use the CFD tools for natural ventilation design.

It is unclear whether window openings and interior layouts influence the flow pattern and pressure distribution around the exterior of a building. If the influence is significant, natural ventilation design requires solving outdoor and indoor airflow simultaneously to optimize window placement and room layout. In principle, it is not a problem to solve the indoor and outdoor airflow at the same time by CFD. However, because of the scale difference between a typical room (meter) and a site plan (hundred meters), a large number of numerical grids must be used to satisfy the necessary resolution requirements. Due to this undesirable complication, this method requires an undue expense to the designer by challenging current PC memory and speed. In many cases, it is almost impossible to evaluate highly complex building designs in this manner with current desktop computer technology. Therefore, this paper attempts to decouple the outdoor and indoor flow simulations by evaluating several simplified methods. The effort is to make the integration of indoor and outdoor airflow simulation a useful design tool that can work on a desktop computer.

RESEARCH APPROACH

One simplified method, named *loose integration* method, is to break up the airflow in and around buildings into two entities: a macroscopic model for the general flow around a site plan and a microscopic model for the flow details within the building. This model cascade is integral to the desired proper flow resolution. Using a site plan and an apartment complex in Shanghai as a model (see Figure 1), this study first examines the feasibility of reintegrating the separate outdoor and indoor simulations for the incorporated design of natural ventilation.

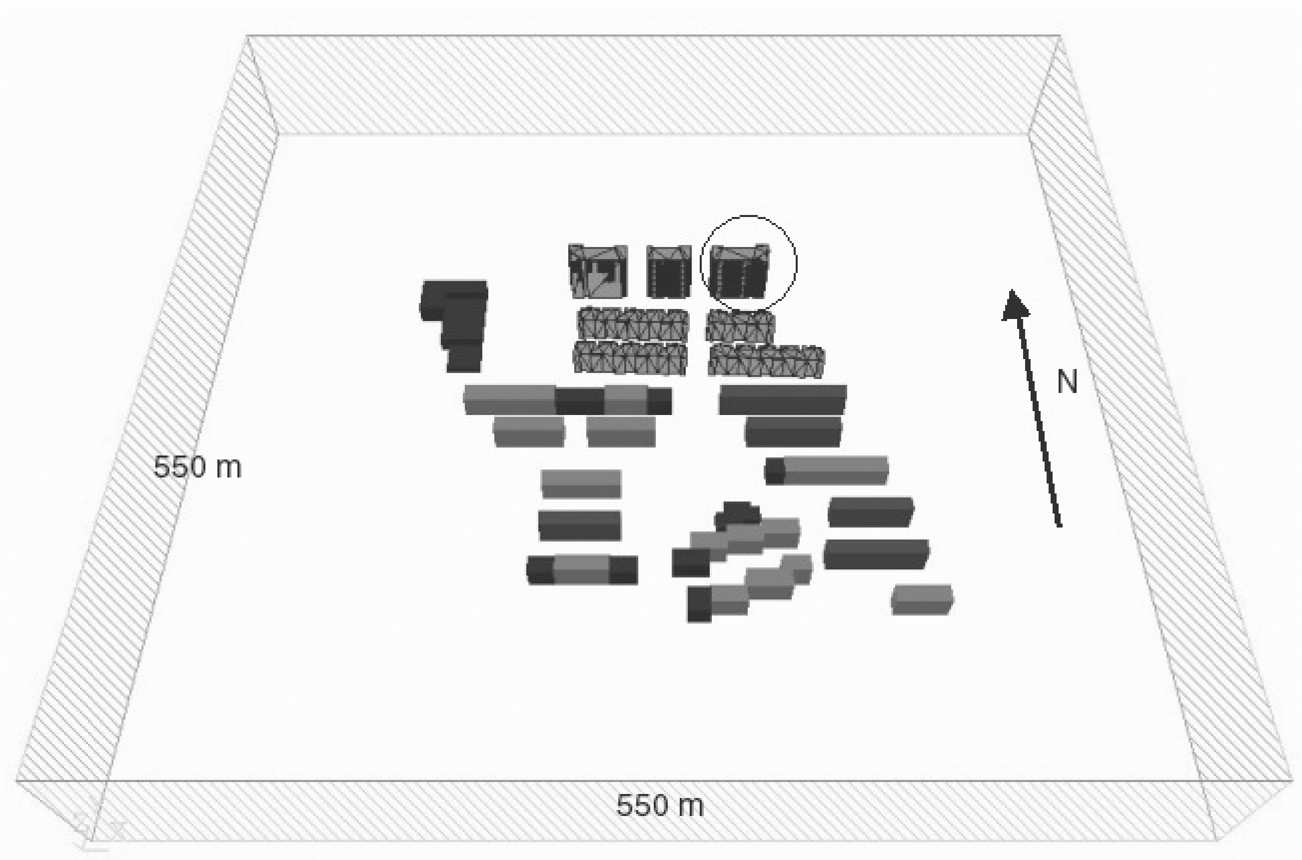

Figure 1: Site plan in Shanghai, China in which the apartment building studied is circled.

The outdoor flow simulation provides the pressure distribution at the windows used in the indoor flow simulations. Under these conditions, indoor airflow for each apartment can be studied separately. This decoupled integration method is based on the assumption that indoor airflow and building openings have little impact on the outdoor airflow and pressure distribution; indoor and outdoor flow fields can therefore be studied separately. This loose integration method completely severs any ties between the outdoor site and an apartment unit, except for the information extracted from the former and used independently in the latter. The outdoor site simulation provides the detailed airflow and pressure information around the building studied. From the apartment simulation, the natural ventilation rate can be determined, and the designers can change the apartment layout and window size and location to maximize natural ventilation.

Due to the influence of building shapes and interference between apartments and other buildings on the site plan, pressure information at particular windows extracted from the loose integration method may not fully guarantee that the corresponded airflow pattern is engaged in a correct and accurate way. The present study carries out a new intermediary method called *compact integration* method, which simultaneously simulates the indoor and outdoor airflow through the apartments of one building with the local outdoor conditions. These outdoor conditions, used as boundary values, are extracted from the site plan simulation, but are only a few meters from the building itself (a detailed explanation follows in the Case Setup section). The large reduction in the domain size allows great detail while still maintaining the predisposition of the airflow observed as it travels across the site.

Both the loose integration and compact integration methods use CFD, a powerful tool for indoor and outdoor airflow simulation. CFD numerically solves the governing equations of fluid flow. These governing equations are derived by applying the principles of conservation of mass and momentum and energy to a control volume of fluid (A thorough treatment may be found in many textbooks on the subject, such as [4]). Because most indoor and outdoor airflows are turbulent, our CFD approach uses turbulence modeling. Turbulence modeling is practical and effective, linking the unknown Reynolds-stresses to the mean flow variable through approximations. The most widely used turbulence model is the "standard" k-• model [5]. In this paper, we adopt a Renormalized-Group k-• model [6] that generally has better accuracy and numerical stability. The flow governing equations are highly non-linear and self-coupled, which make it impossible to obtain analytically exact solutions to most real cases. Therefore in CFD, the equations are solved by discretizing the equations using finite volume techniques that convert them to a set of numerically solvable algebraic equations. The present study uses a commercial CFD software code [7].

CASE SETUP

This study uses a site plan in Shanghai, China to evaluate the research approach methods described above. The evaluated apartment complex, which is 36m high and consists of 12 stories and 48 apartments, is a part of the developing site that includes many different buildings, shown above in Figure 1. Based on the weather data of Shanghai, all the models assume an average wind speed of 3 m/s (at 10 m above the ground) that originates from the southeast. To model the wind profile below 10 m, an exponential function $V = 3\,\mathrm{m/s} \times (\mathrm{Height}/10\mathrm{m})^{0.25}$ incorporates ground roughness in the boundary layer.

Building Model

The present investigation uses a solid block to represent buildings without considering openings in the building façade. This model serves as a basis of comparison for all the methods evaluated in this paper.

The study also uses a hollow shell model to show whether or not building openings have a large effect on the outdoor airflow and pressure pattern, by comparing its results with the solid block model. If the effect of the openings is minimal, a decoupled integration method can be correctly implemented by using the results of the solid block model. The hollow shell model is identical to the solid block model except that the solid building circled in Figure 1 is replaced by the building shown in Figure 2. The modeling of an apartment with windows and without interior partitions is considered to be a worst case scenario, as normally the inclusion of interior walls and furniture greatly increases the interior static pressure by hampering airflow, making the building resemble a solid blockage. However, if the flow and pressure patterns are similarly matched for the blockage and the hollow case, the impact of any interior configuration on the flow will be demonstrated to be minimal. Then one can use the solid block model with confidence.

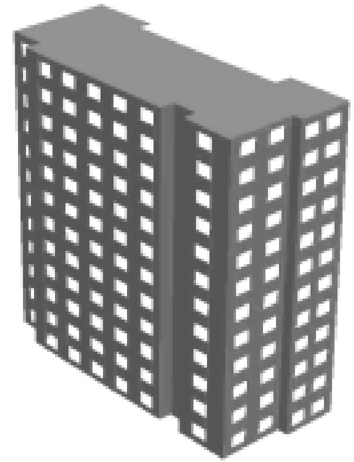

Figure 2: Hollow shell model in CFD, 2m x 2m windows constitute 27% of the vertical wall surface.

Loose Integration Method

The loose integration method separately simulates indoor airflow by assigning fixed boundary condtions at the windows. The boundary values are determined by the pressure values of the solid block model taken at the building façade, halfway up the building. By doing this decoupling, the computing time and capacity can be minimal, compared with that for coupled indoor and outdoor airflow simulation. This method is used to study a typical duplex and single-level apartment in the Shanghai complex. Figure 3 shows the location of the apartments within the building, and Figures 4 and 5 show the floor plans. Furnishings are also added to both apartment units in order to determine how a typical interior configuration affects the airflow.

Compact Integration Method

The compact integration method simulates the interaction of outdoor and indoor flow through and around the apartment building by using boundary values extracted from the solid block model at a set distance from the building. Assuming that it can replicate the outdoor flow of the solid block model, this method will give more accurate indoor airflow results than the loose integration method because it solves outdoor and indoor flow simultaneously.

The greatest challenge in setting up an accurate model is determining the boundary positions and values. Ideally, all pressure and velocity values from the solid block model would be assigned to the boundaries, but this would over-constrain the CFD model because of numerical residuals and precision. Our investigation reveals that fixed pressure and velocity at the upwind east and south boundaries and fixed pressure at the downwind northern and western boundaries, as indicated in Figure 3, replicates the outdoor flow the best. The model is bounded 7m west of the building, 20m south of the building, 20m east of the building, and 20m north of the building; each boundary wall is divided into 25 separate vertical boundary elements. These element values are determined from the solid block model data. The apartment is bounded as a two-story (6m tall) slice.

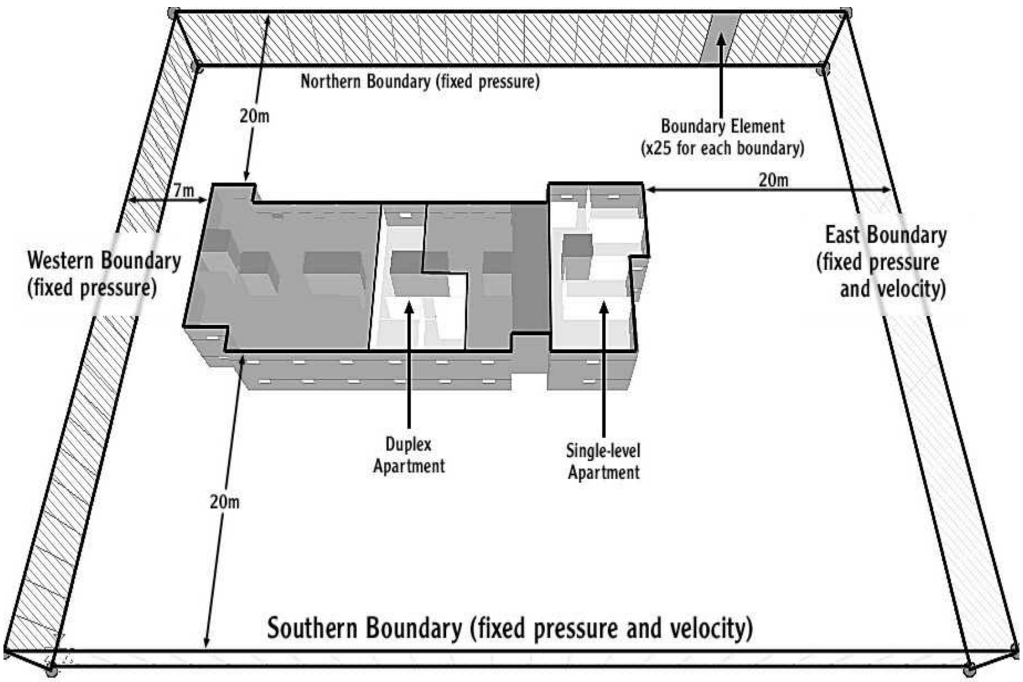

Figure 3: Compact integration method boundary conditions

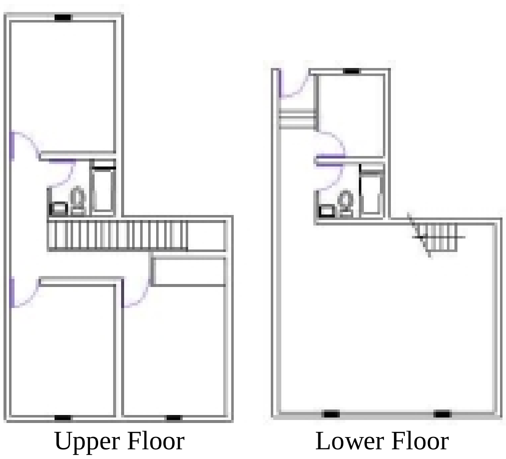

Figure 4: Duplex apartment

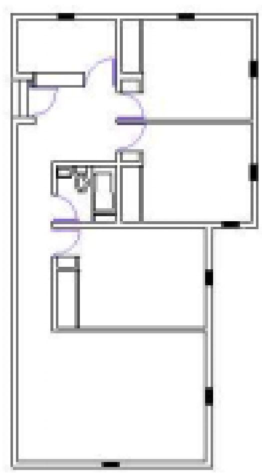

Figure 5: Single-level apartment

RESULTS AND DISCUSSION

Outdoor Airflow

Although modeled on a 550m x 550m site plan (due to the necessary free space around the simulated building group in CFD), the solid block and hollow shell buildings in Figures 6 and 7 show a portion of the localized flow. Similarly, Figure 8 shows a smaller portion of the flow near the building modeled under compact integration. Qualitatively, the three models compare very well for both pressure and airflow distributions. The flow patterns around the building correspond well with each other, with the main difference being the size of the main leeward vortex. The vortex of the hollow model is smaller than the vortices of the solid model and the compact integration model. Also, the separation region on the southwest corner of the building is smaller for the compact integration and hollow shell methods. This is expected since open windows equalize the pressure between the windward and leeward sides and thereby reduce the vortex size. The compact integration method may exhibit errors in the southwest corner because the boundary is relatively close to the building (see Figure 3). Besides these noted exceptions, the flow patterns are quite similar.

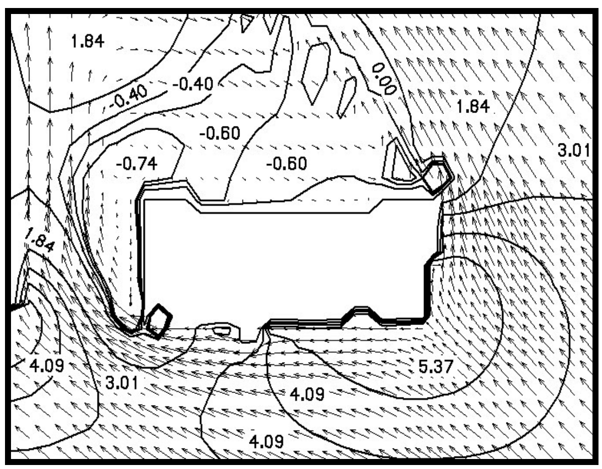

Figure 6: Solid block site plan flowfield and pressure contour

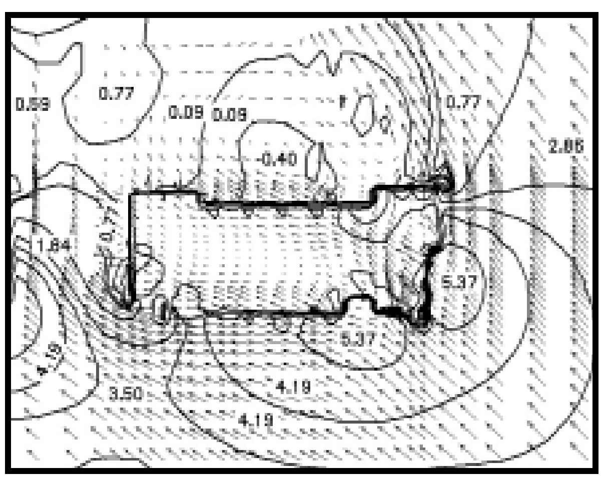

Figure 7: Hollow shell site plan flowfield and pressure contour

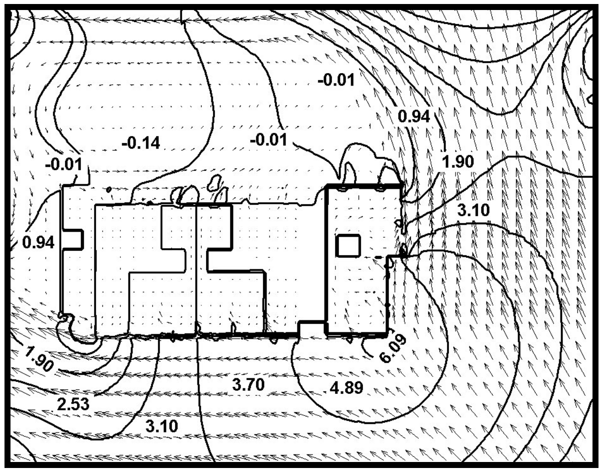

Figure 8: Compact integration method flowfield and pressure contour

There are some disparities in the hollow and compact integration models when comparing pressure values with the solid block model. Figure 9 shows the pressure distributions of each model along the north and south walls of the building. Table 1 shows the average of the pressure values along all four edges of the building. The pressure and airflow data for the three models are taken at a plane 18m above the ground (halfway up the building) that is centered vertically at a row of windows. It may be important to remember that the wind originates from the southeast, and is relatively unobstructed by buildings downwind.

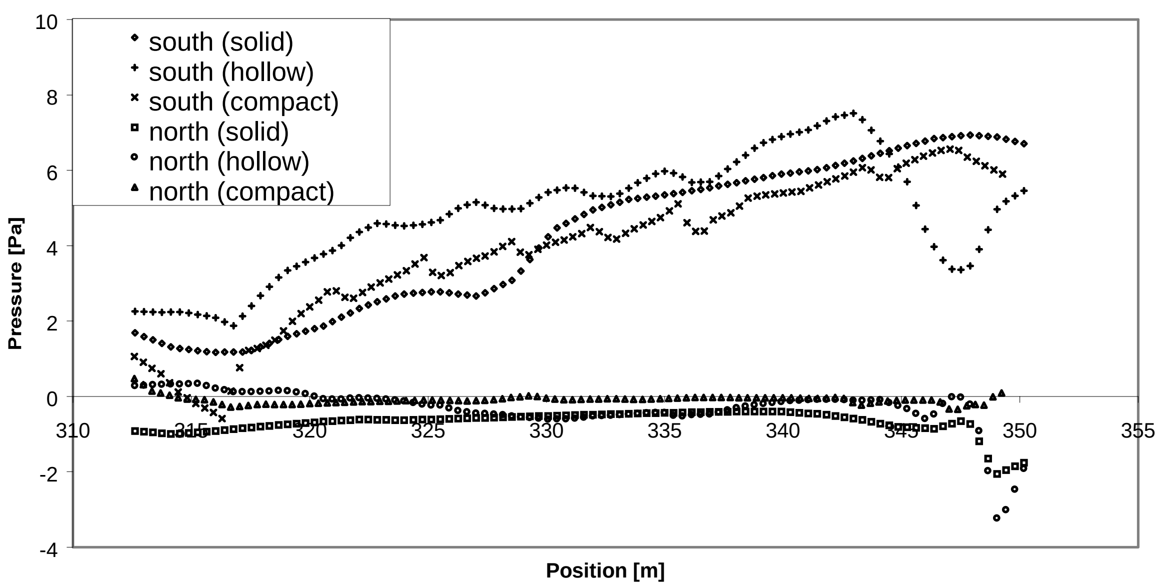

Figure 9: North and south face pressure extracted from three models

Table 1: Average relative pressures along each building face for the three models

	North	South	East	West
Solid-model	-0.48 Pa	4.25 Pa	5.31 Pa	-0.79 Pa
Hollow-model	-0.31 Pa	4.81 Pa	5.22 Pa	0.07 Pa
Compact Integration Method	-0.09 Pa	3.84 Pa	3.70 Pa	1.02 Pa

On the north side, all the pressures are within 0.39 Pa for the three cases. Any disparities in pressure on the north side are caused by the presence of windows. The solid model dutifully diverts the main wind into two parts, one that flows parallel to the south face, and one that flows parallel to the east face. At the northeast and southwest corners of the solid model, it is clear that separation occurs, thus producing a large negative pressure (Figure 6). However, due to the windows in the hollow case, the flow is diverted many times, along the south and east face through the windows (Figure 7). Thus, less wind passes parallel to the east side, generating a smaller separation and thus a smaller low-pressure region at the northeast and southwest corners. For the compact integration method, the problem is magnified since it is missing the vortex at the southwest corner because the boundary is relatively close to the wall.

Large differences occur on the north and west sides, where the pressures are relatively close to zero, so minor fluctuations or disparities in the pressure comparison are more dramatically represented. Especially when either model shows a completely different pressure from the other (e.g. all of the pressures on the west face of the solid model are negative, while more than half pressures from the hollow model are positive), dissimilarities in the comparison of the values become extremely marked.

The most important value of pressure used for the simulations is the gradient between the north and south faces, as this is the main direction of flow in the apartments. Compared to the pressure gradient of the solid-block model, the average difference in the pressure gradient for the hollow shell model is 5.8% while that of the compact integration model is 16.8%. It is thus reasonable to conclude from the hollow shell case that room partitions and windows do no contribute to a major difference in flow pattern or pressure field, and that values from the solid block case can be used as boundaries for decoupled models. The compact integration method exhibits some notable differences in its pressure distributions, most likely caused by the method of setting its boundaries. However, the qualitative similarities of all three methods and the relatively low quantitative error suggest that the compact integration method will successfully predict the outdoor and indoor airflow.

Indoor Airflow

This part of the study determines the accuracy of simulating indoor airflow using both loose and compact integration methods with the outdoor flow study. The compact integration method evaluates the same duplex and single-level apartment layouts as the loose integration method, connecting the indoor flow with the localized outdoor flow. The resulting airflow patterns and air change rates for the loose and compact integration methods are compared. If the indoor airflow results from both integration methods are found to be compatible, the loose integration method can then be applied, decoupling indoor and outdoor airflow completely. This reduction in computing time allows the designer to more easily modify the apartment and window layout in order to maximize natural ventilation.

Figures 10 and 11 show the indoor airflow patterns (through a horizontal plane at the center height of the windows) resulting from the loose and compact integration methods, respectively. Both methods show flow entering the same windows and exiting the same windows of the apartments, and the overall flow path within the apartments is similar.

Though the overall flow path through each apartment is similar, localized airflow patterns are different, especially at the windows. The loose integration method shows airflow distributed evenly in all directions as it enters the windows, because it has fixed-pressure boundaries at the windows and no set momentum. The compact integration method, however, shows highly directional velocity vectors entering and exiting the windows, which more accurately depicts the real situation. The directionality of the velocity vectors creates more vortices and recirculation inside each room for the compact integration method. The southern section of the duplex apartment (Figure 10) illustrates one such area of recirculation.

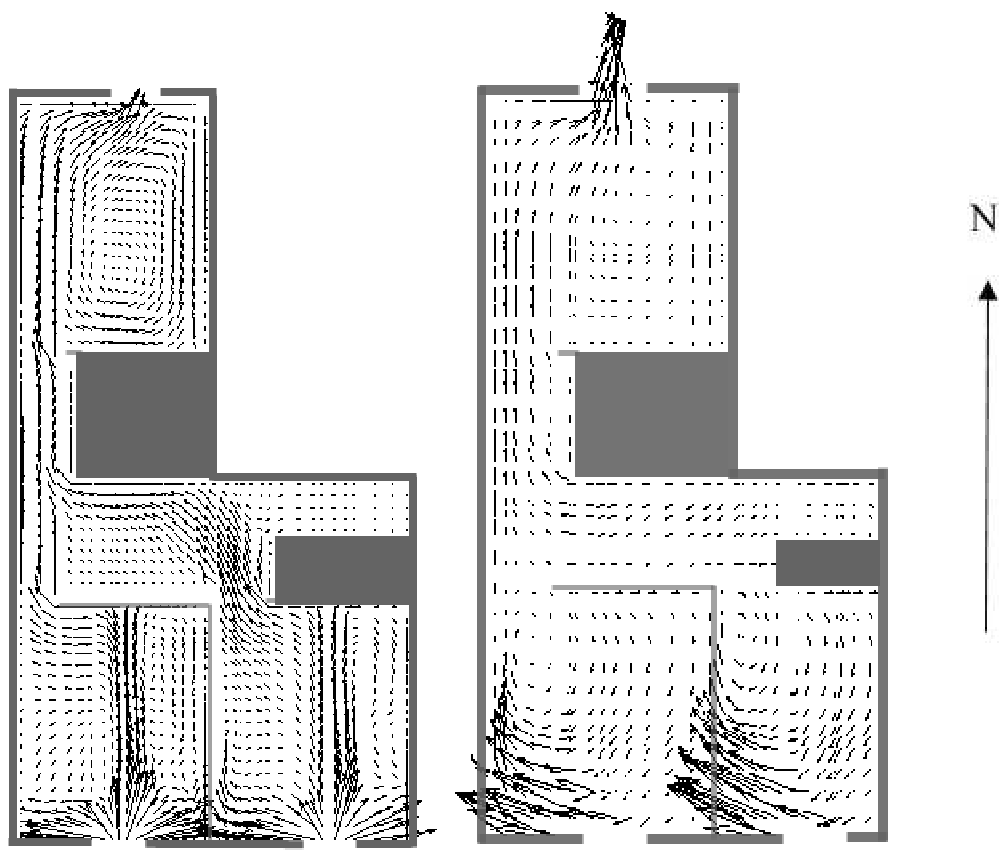

Figure 10: Comparison of velocity vectors at the center height of the windows for the upper floor of the duplex apartment (the lower floor has similar results) – loose and compact integration methods

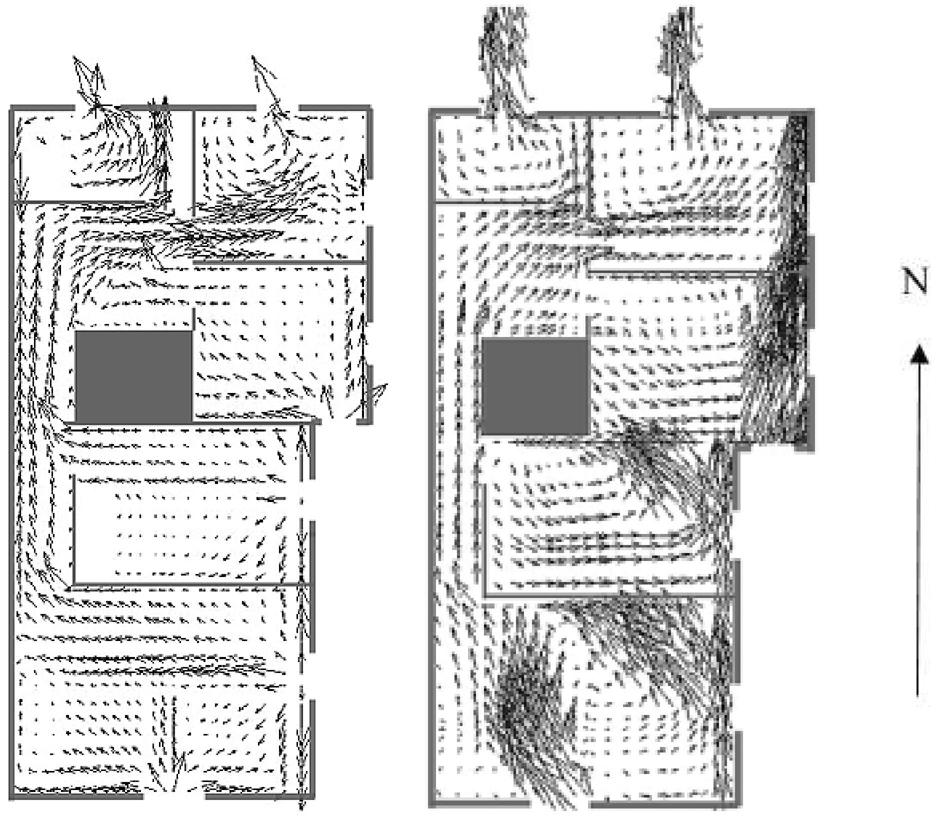

Figure 11: Comparison of velocity vectors at the center height of the windows for the single-level apartment – loose and compact integration methods

Calculating the airflow rate through each apartment provides a quantitative comparison of the indoor airflow results. Table 2 compares these results in air changes per hour (ACH). Results obtained from a simplified hand-calculation method are also presented in the table. This hand-calculation method converts a pressure gradient into velocities by using a discharge coefficient of 0.6 through the windows and doorways of the apartments.

Table 2: Comparison of air change rates for duplex and single-level apartments using three methods.

	Loose Integration Method	Compact Integration Method	Hand Calculation
Single-level apartment	38 ACH	36 ACH	27 ACH
Duplex apartment	19 ACH	18 ACH	17 ACH

Compared to the airflow rates of the loose integration method, the differences in airflow rates for the compact integration method are 5% for both the duplex and single-level apartment. It can therefore be concluded that these two methods are similar and the loose integration method will accurately predict the air change rate with less computing power. However, the compact integration method will provide more accurate flow patterns that may be important when determining occupant comfort due to wind velocity.

It is interesting to note that the hand calculation method successfully predicts the ACH for the duplex apartment well, but not for the single level apartment. The location of the single-level apartment at the corner of the building produces complex two-dimensional flow, but the duplex (located in the center of the building) has roughly one-dimensional flow that hand calculations can easily predict.

CONCLUSIONS

This paper develops the loose integration and compact integration methods for the calculation of natural ventilation through large openings with CFD. The methods decouple the indoor and outdoor airflows to reduce the required computing power. Further, a comparison of a solid block building model with a hollow building model reveals that the windows on the building façade have little impact on the pressure distribution around the building.

Building designers can use the loose integration method for calculating airflow rate through a naturally ventilated building, despite its error in predicting the airflow pattern near windows. The compact integration method can provide more accurate details of the indoor and outdoor airflow patterns.

In comparing the differences between the data extracted from a solid and hollow building, it is apparent that there are some palpable disparities between the pressures calculated at the building façade faces. However, it must be stressed that the pressure difference between the windward and leeward sides has a cumulative error of less than 6%, which is acceptable value for design practice. Since the simulation of individual apartment plans ultimately uses a pressure difference to induce flow, the technique of the hollow model simulation is superfluous and ultimately unnecessary. Thus, the time between iterations of design evaluations is greatly shortened by the reduction of an extra step.

In all, to optimize natural ventilation, these methods will promote predictive and iterative design solutions. As all the boundary conditions are solved previously on a macroscopic scale, the bulk of the computing time is on the microscopic refinement, not on the recalculation of flow far from the building.

ACKNOWLEDGEMENT

This work is partially supported by the U.S. National Science Foundation under grant CMS-9877118.

REFERENCES

~~1~~1.Carrilho da Graca G., Chen Q., Glicksman L.R., and Norford L.K.
"Simulation of Wind Driven Ventilative Cooling in an Apartment Building in Beijing and Shanghai"
Proceedings of the 3rd International Symposium on HVAC, ISHVAC'99, Vol.2, 1999, November 17-19, Shenzhen, China, pp648-658.

~~3~~2.Li Y., Delsante A. and Symons, J.
"Prediction of Natural Ventilation in Buildings with Large Openings"
Building & Environment, Vol.35, 2000, pp191-206.

3.Ayad S.S.
"Computational Study of Natural Ventilation"
Journal of Wind Engineering & Industrial Aerodynamics, Vol.82, 1999, pp49-68.

~~4~~4.Anderson D.A., Tannehill J.C., and Pletcher R.H.
"Computational Fluid Mechanics and Heat Transfer"
Hemisphere Publishing Company, New York, 1984.

~~5~~5.Launder B.E. and Spalding D.B.
"The Numerical Computation of Turbulent Flows"
Computer Methods in Applied Mechanics and Engineering, Vol.3, 1974, pp269-289.

6.Yakhot V., Orzag S.A., Thangam S., Gatski T.B., and Speziak C.G.
"Development of Turbulence Models for Shear Flows by a Double Expansion Technique"
Physics Fluids A, Vol.4, 1992, pp1510-1520.

7.CHAM
"PHOENICS Version 3.1"
CHAM Ltd., London, 1999.