ELSEVIER

A comparison between CFD and Network models for predicting wind-driven ventilation in buildings

Omar S. Asfour*, Mohamed B. Gadi

School of the Built Environment, The University of Nottingham, University Park, Nottingham NG7 2RD, UK

Abstract

The aim of this paper is to compare the implementation of computational fluid dynamics (CFD) and Network models for airflow rate estimation in buildings. The CFD software used is Fluent 5.5. Comparison between the predicted and simulated airflow rate is suggested as a validation method of the implemented CFD code, while the common practice is to compare CFD outputs to wind tunnel or full-scale measurements. This could be useful for studies that have no access to laboratory or full-scale testing facilities. Results obtained from testing a number of cases have been compared and analysed, considering normal and oblique wind directions. The comparison held between mathematical and CFD results generally showed a good agreement, which seems to justify the use of CFD code for predicting natural ventilation in buildings.
© 2007 Elsevier Ltd. All rights reserved.

Keywords: Natural ventilation; CFD; Network model; Validation

1. Introduction

Application of computational fluid dynamics (CFD) was originally introduced for industrial applications, but today it has also become a common tool for assessing building ventilation and environmental performance. As described by CIBSE [1], CFD is "a very powerful technique" in predicting air movement and characteristics. CFD model is based on the concept of dividing the solution domain into sub-zones. Then, for each zone, the mass, momentum, and energy conservation equations are solved, utilising the processing power of computers. This helps to perform calculations more easily and, in comparison with natural ventilation mathematical models, gives more detailed results. For example, CFD codes are used to predict airflow rate, air velocity and temperature, and airflow patterns inside and around buildings. Many softwares based on CFD codes have been developed like, Fluent, Phoenics, and Flovent.

In the last few years, an intensive work has been done using CFD. Comparisons of CFD results with wind tunnel tests have shown good agreement. However, some studies are limited to the use of CFD modelling, and have no access to experimental testing facilities. In this case, it is crucial to validate the implemented CFD code prior to the proposed modelling study in order to avoid producing any misleading results. This is because CFD is considered to be a sophisticated modelling technique. One possible option is to compare CFD results to natural ventilation mathematical models, which is the scope of this paper. These models vary from simple to complicated models. The commonly used Network mathematical model has been found to be appropriate for this study, where calculation process is reasonable.

Both implemented CFD and Network models are briefly explained below. This followed by the results obtained from both models regarding airflow rate estimation in the case of normal and oblique wind direction.

2. Network mathematical model

A flow network is used here to represent different airflow paths in buildings. This network is represented by a group

---

*Corresponding author. Tel.: +44 115 9513118; fax: +44 115 9513159.

E-mail addresses: laxosa@nottingham.ac.uk, o.asfour@hotmail.com (O.S. Asfour), mohamed.gadi@nottingham.ac.uk (M.B. Gadi).

of nodes, simulating building openings, and a group of lines, simulating flow paths. Thus, it has more flexibility in analysing natural ventilation problems. Bahadori and Haghighat [2] explained the implementation of this model. Assuming a multi-zone problem, i.e., a building with N zones, a number of N equations will be established in the Network model. Each equation is dependant on the previous one.

Thus, these equations are solved iteratively, where the summation of mass flow rates should be zero. In the case of wind-induced ventilation, the knowledge of internal and external pressure coefficients for each opening is required to find out pressure difference across this opening. These coefficients are usually obtained experimentally or from standard pressure coefficients data. The following equation is commonly used in predicting airflow rate induced by wind pressure difference:

$$Q_n = A_{\text{eff}} \sqrt{(2\Delta p/\rho)}, \qquad (1)$$

where Q_n is airflow rate through an opening n (m³/s), A_{eff} is the effective area of this opening, ΔP is pressure difference across it (Pa), and ρ is air density (kg/m³). Pressure difference mentioned in the previous equation can be estimated using

$$\Delta P = 0.5\rho V^2 \left| (C_{pn} - C_{pi}) \right|, \qquad (2)$$

where V is wind velocity at datum level (m/s), C_{pn} is pressure coefficient at opening n, and C_{pi} is pressure coefficient inside the space. Substituting Eq. (1) into Eq. (2), we get

$$Q_n = A_{\text{eff}} V (C_{pn} - C_{pi}) \left| (C_{pn} - C_{pi}) \right|^{-1/2}. \qquad (3)$$

Considering the Law of Mass Conservation, this equation can be rewritten as

$$\sum_{n=1}^{N} A_{\text{eff}} V (C_{pn} - C_{pi}) \left| (C_{pn} - C_{pi}) \right|^{-1/2} = 0. \qquad (4)$$

Therefore, it is in possible to estimate internal pressure coefficient using Eq. (4) and then airflow rate using Eq. (3). To do so, the knowledge of C_{pn} is essential, as discussed in Section 4.

3. CFD code

Fluent 5.5 program, one of the most widely used commercial CFD codes, has been used in this study. Gambit 1.3 program, which is a pre-processor program, has been used to define the building geometry. In fact, using two-dimensional modelling in room ventilation problems does not give realistic simulation of airflow, as it does not consider some phenomena that determine airflow characteristics, such as airflow separation over building sharp edges. Thus, the use of the three-dimensional modeling has been chosen in this study. The disadvantage, however, is that this method significantly increases the required time for solution convergence.

Boundary and continuum types have been also defined in Gambit program, where velocity inlet and outflow boundary types have been used for the solution domain. In order to predict airflow behaviour in the modelled buildings, CFD calculation mesh has been generated, which replaces the air inside and around the building. The idea of this mesh is to divide the solution domain into small cells, which are used to predict airflow behaviour using computer processing ability. This can be achieved using different types of mesh, like hexagonal or tetrahedral.

For these cells, or volumes in the case of three-dimensional modelling, Fluent 5.5 software can numerically solve the three basic conservation equations of mass, momentum and energy in an iterative manner. However, wind-induced ventilation is believed to be more effective in hot climates, when compared with stack-induced one. This is because of the relatively lower difference between indoor and outdoor temperatures, which is the main factor affecting stack ventilation [3]. Therefore, energy settings have been set off in this study.

To process these calculations, an appropriate mathematical model has to be applied on the solution domain. Most of airflow problems in buildings consider airflow to be turbulent. Thus, definition of turbulent model is required to help solving the transport equations. Awbi [4] mentioned that the standard k–ε Model is believed to be the most used and developed turbulence model. This model is most likely to predict reasonable results for airflow studies in buildings. Turbulence characteristics have been specified using the Turbulence Intensity and Length Scale option, which is recommended in room airflow problems, and depends on the Reynolds number and the inlet size of the case. Segregated solver has been used, where the fundamental equations are solved sequentially or segregated from each other. The solution reaches the end when the convergence criteria are met.

The modelled prototype represents a room with an inlet and an outlet. In Fluent 5.5 program, it is required to define air velocity magnitude at this inlet. Air velocity magnitude varies with height. Thus, it is crucial to estimate air velocity at building height, as will be discussed in Section 5. Rotating the building model inside the three-dimensional solution domain, which represents the ambient air, can be used to simulate wind angle. It is important to note that Guage pressure should be set to 0. This is because the only pressure acts is the atmospheric pressure. The solution progresses in the form of several iterations, which can be monitored by the user. During this iteration process, the residual information of the velocity, continuity, and the turbulence parameters of the viscous model are continuously updated. Convergence is achieved when a sufficient error tolerance, defined by the user, is reached. Many outputs in different presentation methods can be obtained from Fluent 5.5 software. This includes airflow rates and contours of air velocity magnitudes.

4. Prediction of airflow rate using Network mathematical model

It is possible to implement this model for any opening knowing its area and wind static pressure coefficient at this opening. Prediction process here is divided into four stages:

- specification of pressure coefficient data;
- specification of building configurations;
- calculations of internal pressure coefficient;
- calculations of airflow rate.

Pressure coefficients data that will be used here are those recommended by Liddament [5]. These data have been produced in wind tunnels for some common building configurations, and are considered more developed than the data mentioned in the British Standards [6]. This is in terms of testing more wind angles and considering the effect of terrain nature. Cases proposed here have nearly the same volume, but different aspect ratios. Aspect ratios of 1:1, 1:2 and 2:1 are considered, as depicted in Fig. 1. This figure also shows values of pressure coefficient, with a reference number indicating wall number. Initially, calculations have been performed considering a relatively low and high reference wind velocities, namely 1 and 5 m/s, and two wind angles, namely 0° and 45°. However, it has been observed that discrepancy percentage is not sensitive to wind velocity magnitude, as the percentage is nearly the same in the case of same building geometry and different approaching wind speeds. This result supports what Liddament [5] has mentioned that pressure coefficient is normally assumed to be independent of wind speed but not direction. Thus, the following investigation will be confined to velocity magnitude of 1 m/s.

It is important to note that at oblique wind direction, effective area of the windows is less. By knowing that wind angle is 45°, window effective width can be simply calculated using Right Triangle Trigonometry. Thus, window effective area can be estimated by multiplying window width by its height. This is $1.4 \times 2 = 2.8 \, \text{m}^2$. Internal pressure coefficients can be estimated from Eq. (4). As the above-illustrated models have only two zones: indoor and outdoor, this equation can be simplified as

$$\left| (C_{pn} - C_{pi}) \right|^{-1/2} + \left| (C_{p(n+1)} - C_{pi}) \right|^{-1/2} = 0. \tag{5}$$

It is required to know air velocity at building height to implement Network model for airflow rate estimation. This is possible using the following common equation [7]:

$$V = V_r c H^a, \tag{6}$$

where V is wind speed at datum level (m/s), V_r is reference wind speed (obtained from meteorological data) (m/s), H is the height of the building, c is parameter relating wind speed to terrain nature (0.68 in the open country terrain), and a is an exponent relating wind speed to the height above the ground (0.17 in the open country terrain). Knowing different building heights, as illustrated in
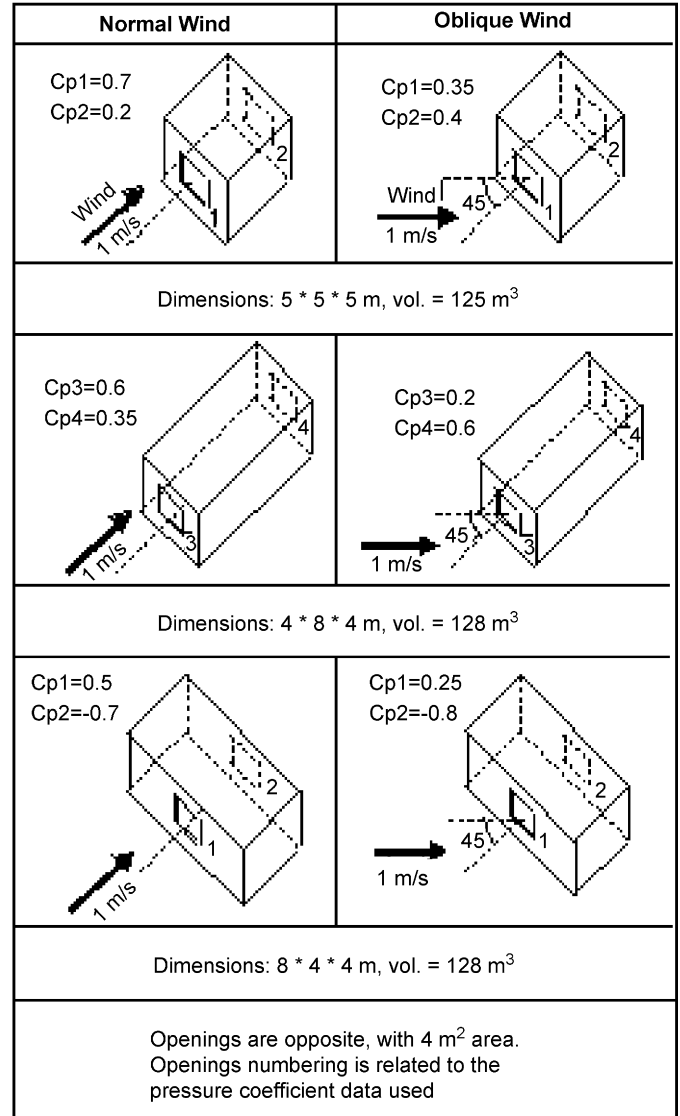

Fig. 1. Illustration of the cases tested in this study.

Table 1
Airflow rate prediction, using the Network model

Case	Opening	A_{eff} (m²)	C_{pi}	Q_n (m³/s)	Q_n (kg/s)
1	1	4	0.25	2.40	2.94
	2	4		−2.40	−2.94
2	3	4	0.125	2.37	2.90
	4	4		−2.37	−2.90
3	1	4	−0.1	2.66	3.26
	2	4		−2.66	−3.26
4	1	2.8	−0.025	1.53	1.88
	2	2.8		−1.53	−1.88
5	3	2.8	−0.2	1.52	1.87
	4	2.8		−1.52	−1.87
6	1	2.8	−0.275	1.75	2.14
	2	2.8		−1.75	−2.14

Table 1, wind velocity at building height has been estimated and implemented in Eq. (3) to estimate airflow rate. Results are shown in Table 1.

This airflow rate is given in m^3/s. It is possible to convert it to mass flow rate, in kg/s, by multiplying it by air density in order to facilitate the comparison with CFD results. Value of air density should match the one under which pressure coefficient data were estimated. As there is no indication of this value in the related resource, this value will be assumed to be $1.225 \, kg/m^3$, which is the default value used in Fluent 5.5 program and nearly the same of the standard air density value, i.e., $1.2 \, kg/m^3$.

5. Prediction of airflow rate using CFD modelling

Using Fluent 5.5 program, it is possible to estimate airflow rate through building openings. Modelling process includes the following four stages:

- drawing of building models in Gambit program;
- generating and exporting the calculation mesh to Fluent 5.5 program;
- definition of solution code;
- computing airflow rate.

As discussed in Section 3, building models are simulated three-dimensionally in order to obtain more realistic airflow pattern. This explains the significant differences between results obtained using both two and three-dimensional modelling in the early stage of this study. Thus, it is important to simulate the ambient air around the building. To do so, the modelled room will be placed inside a three-dimensional box, as shown in Fig. 2.

By rotating the room inside this box, it is possible to simulate oblique wind direction. However, this requires a larger domain to ensure solution convergence. For example, in the case of $0°$ wind direction, the building is placed in a $30 \, m \times 30 \, m \times 20 \, m$ box. In $45°$ wind direction, the extra sheer stresses, resulted by the oblique building walls, cause reversed flows to occur. This leads the solution to diverge, and makes it essential to use a larger domain size. This size has been gradually increased until an acceptable size of $50 \, m \times 50 \, m \times 20 \, m$ was achieved (Fig. 3).

In ventilation modelling, it is important to consider wind speed variation with height, due to the frictional effect of the ground. It is possible to do so using Eq. (6). Wind velocity profile can be defined along the velocity-inlet, as illustrated in Fig. 2, using 'User Defined Function' option in Fluent 5.5 software. However, application of this option
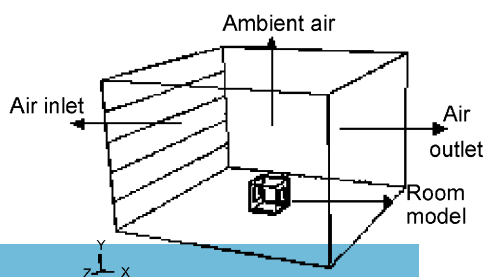

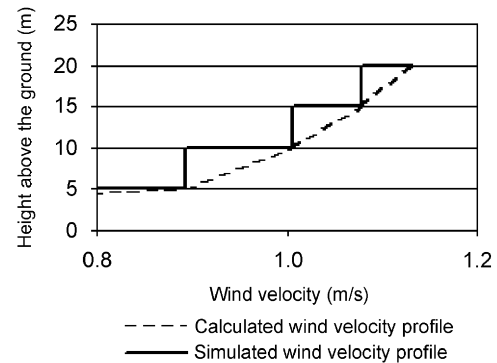Fig. 2. Simulation of the ambient air around buildings.

Fig. 3. Illustration of the calculated and simulated wind velocity profile (for case 1, as an example).

Table 2
Air velocities for the different sub-inlets used in CFD modelling, as estimated from Eq. (6)

Height above the ground (m)	Sub-inlet number	Wind velocity (m/s)
Cases 1 and 4 (room height is 5 m)		
5	1	0.894
10	2	1.006
15	3	1.078
20	4	1.132
Cases 2,3,5 and 6 (room height is 4 m)		
4	1	0.861
8	2	0.968
12	3	1.037
16	4	1.089
20	5	1.132

in three-dimensional simulation required some advanced C^{++} programming. It is possible, as an approximation method, to divide the velocity inlet into many sub-inlets. Each sub-inlet will have a different air velocity magnitude depending on its height above the ground. This method has been found to be useful and good results have been obtained, as has been concluded from this study. To allow for the estimation of airflow rate, it is recommended to split the solution domain into the following zones: room walls, ambient air, room interior, and openings volumes. The last three volumes were defined as fluid continuum. The benefit of this arrangement is that Fluent 5.5 program will define openings surfaces separately, which facilitates airflow computing process (Table 2).

The use of three-dimensional simulation leads to think about the resulting mesh sizes and the required processor capacity. For example, the use of 0.2 m mesh spacing for the entire domain, $30 \, cm \times 30 \, m \times 20 \, m$ in cases 1–3, was beyond computer memory and speed in this study. A common solution here is to create a hierarchy in mesh size to be fine inside the building and larger around it. In any case, a trial-and-error process is recommended to find out the most appropriate mesh configuration. Rough meshes can be suitable and sufficient in many CFD simulation

cases [8]. Generally, the following mesh configuration has been found to be acceptable:

- *Meshing scheme*: hex-map or hex-submap. Mesh elements here are hexahedral, which is applicable to the shape and topological characteristics of the modelled cases. In case of oblique wind, the use of tetrahedral mesh has been found to give more accurate results, as discussed in the following section.
- *Mesh node spacing*: in case of normal wind direction, this spacing is 0.6 m. In the case of oblique wind direction, solution domain was divided into two zones: room interior, with 0.5 m spacing in the hex-map mesh, and room exterior, with 1 m spacing in the tetrahedral mesh.

Once the mesh is ready, it can be exported to Fluent 5.5 program in order to perform the calculations, which may take several hours depending on the size and complexity of each individual case. Fig. 4 illustrates the resulting airflow pattern for these cases, presented by contours of velocity magnitudes over a 2-m height section. Generally, the

observed airflow pattern, in all the cases, has been found to be reasonable for such sharp-edge geometries. For example, when wind reaches the windward face, a high-pressure zone is formed there. This pressure pushes air inside, around, and over the building. Some standard features can also be observed. This includes airflow separation over building sharp edges. This phenomenon usually occurs when airflow layers hit a sharp edge of the building and thus lose their momentum. After some distance, the separated airflow joins its original stream again in a point called the reattachment point.

After achieving solution convergence, it is possible to obtain the mass flow rate directly from the software, utilising the Surface Integrals option for the relevant inlet surface. Results obtained for the different cases are illustrated in Table 3.

6. Comparison between airflow rate prediction using Network and CFD models

Table 4 shows a comparison between airflow rate predicted by the Network and CFD models. The negative
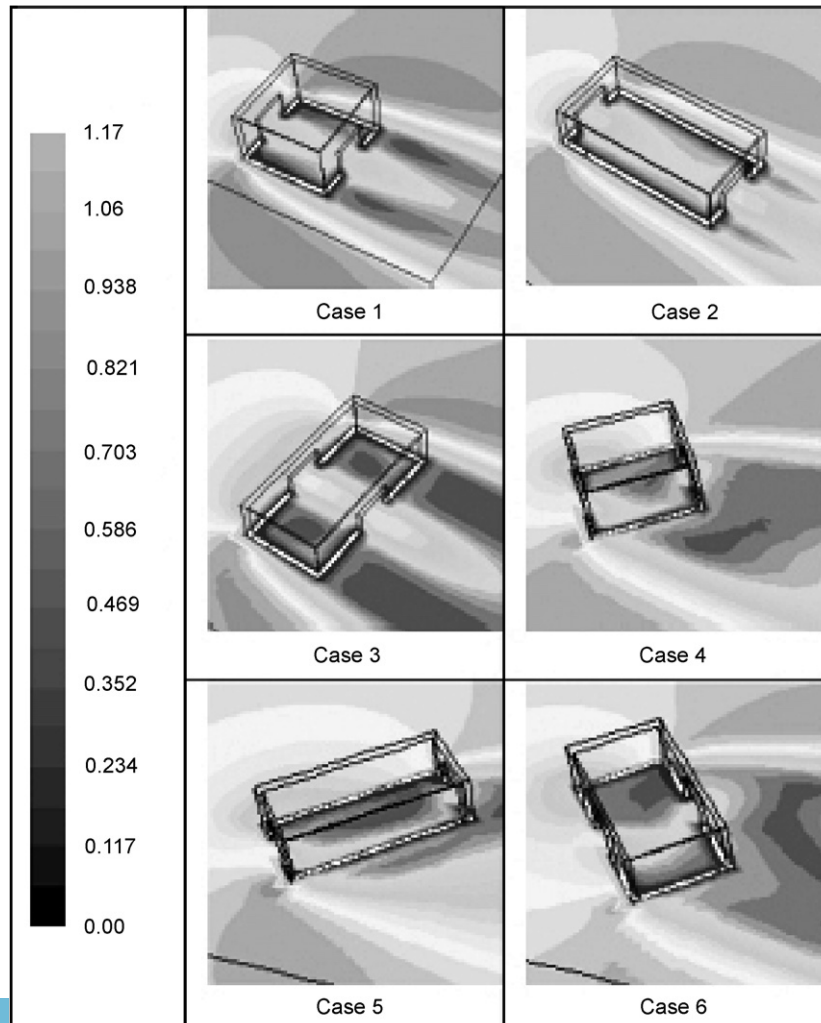

Fig. 4. Airflow pattern, presented by contours of velocity magnitudes, for the cases modelled in this study.

Table 3
Airflow rate prediction, using CFD model

Case	Opening	Q_n (kg/s)	Case	Opening	Q_n (kg/s)
1	1	3.19	4	1	1.78
	2	−3.19		2	−1.78
2	3	2.98	5	3	1.69
	4	−2.98		4	−1.69
3	1	3.30	6	1	1.92
	2	−3.30		2	−1.92

Table 4
Discrepancy percentage between estimated and modelled airflow rate for cases 1–6

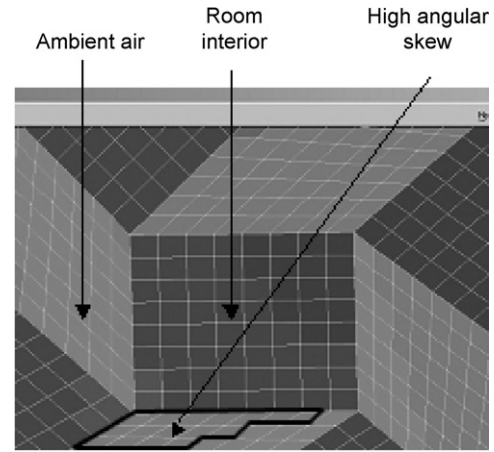| Case	Opening	Q_n (kg/s) (CFD)	Q_n (kg/s) (Network)	Discrepancy (%)
1	1	3.19	2.94	7.8
	2	−3.19	−2.94	
2	3	2.98	2.90	2.7
	4	−2.98	−2.90	
3	1	3.30	3.26	1.2
	2	−3.30	−3.26	
4	1	1.78	1.88	−5.6
	2	−1.78	−1.88	
5	3	1.69	1.87	−10.7
	4	−1.69	−1.87	
6	1	1.92	2.14	−11.5
	2	−1.92	−2.14	

Fig. 5. Skew observed in the hex-map and sub-map meshes used in the case of 45° wind direction.

Table 5
Discrepancy percentage between estimated and modelled airflow rate in the case of oblique wind direction, after changing mesh type

Case	Opening	Q_n (kg/s) (CFD)	Q_n (kg/s) (Network)	Discrepancy (%)
4	1	1.84	1.88	−2.2
	2	−1.84	−1.88	
5	3	1.83	1.87	−2.2
	4	−1.83	−1.87	
6	1	2.26	2.14	5.3
	2	−2.26	−2.14	

sign indicates that the modelled airflow rate is less than the calculated one. In general, it shows that a good agreement has been achieved. Discrepancy percentage observed is usually acceptable in airflow rate prediction, which is given as a snapshot and measured in kg/s.

However, discrepancy percentage, in general, is higher in the case of oblique wind direction. CFD code used here is the same of that one used with the normal wind direction. However, it has been found that the use of hex-map and sub-amp meshes in the oblique wind direction results in a less mesh quality. This is because tilted walls resulted in high angular skew between the edges of mesh cells, about 0.7, which is considered high. For example, an excellent mesh has a skew less than 0.25 [9] (Fig. 5).

Therefore, mesh type of the ambient air zone has been changed to the tetrahedral mesh, which has more flexibility in meshing such geometries. Hex-map mesh is still used for room interior. Table 5 shows a recalculation of the comparison held between results obtained from both mathematical and CFD models in the case of oblique wind direction. Change of mesh type seems to have a good effect, as discrepancy percentage has been significantly reduced in all the three cases.

The differences observed between airflow rates predicted by the Network and CFD models at both wind directions can be justified by many reasons. One of them is the approximation method used in simulating air velocity

profile, as explained in Section 5. This is because wind-induced airflow rate is dependant on wind velocity. As the square of air velocity is used in the estimation of this pressure difference, any error in air velocity results in a larger error in airflow rate value.

Another reason can be the approximation of the mathematical procedure used. This is, on one hand, because the used wind pressure coefficient values are averaged over the whole specified building face, and not a specified position on it. Liddament [5] highlighted this point and told that accurate evaluation of this parameter (i.e. pressure coefficient) is one of the most difficult aspects of air infiltration modelling. In addition, these data were generated in wind tunnel experiments, where air density, and therefore pressure, is affected by air temperature, which is not the case in the isothermal CFD simulation carried out. On the other hand, air infiltration model used contains many assumptions to enable the estimation of airflow rate through a reasonable mathematical process. On the opposite, CFD considers the different values of air pressure on the opening, and calculate airflow rate as a summation their product.

One more reason is related to the pressure coefficient data used in the case of oblique wind direction. Air pressure distribution around a solid model changes if it is provided with openings. In the case of 45° wind incidence,
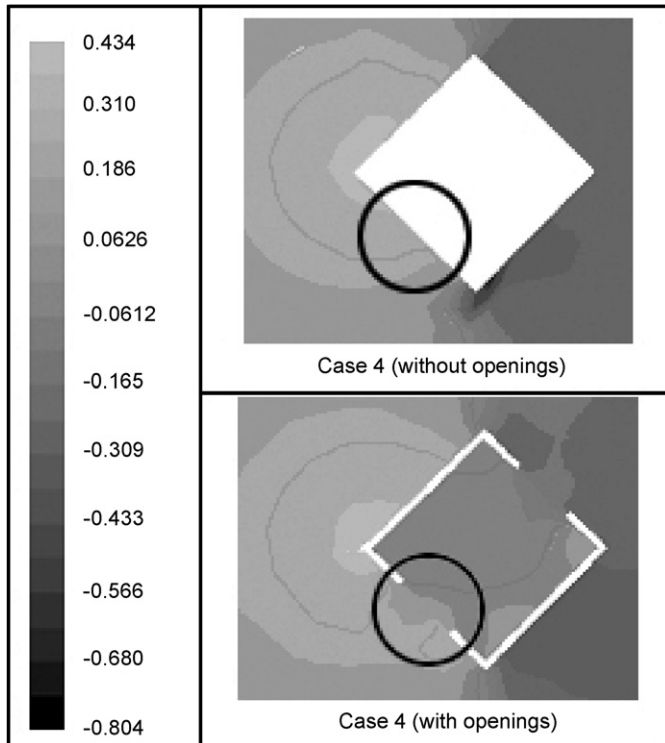
Fig. 6. Contours of pressure magnitude (Pa) showing different pressure distributions in the modelled cases and the standard solid ones.

building model has two windward faces. In the case of a solid model, average pressure coefficient over these windward faces is the same for the square cases, and has a relatively small difference in the rectangular one. In the case of placing an opening at one of these two windward faces, it is expected that air pressure distribution will change, and there will be no more balance at its distribution on these windward faces. This is because windward with solid geometry receives more wind deflection on it. On the opposite, windward with an opening receives less wind deflection, and therefore wind pressure will be less too. For example, pressure coefficient values on the windward faces in case of 45° wind direction and a square building form are the same, i.e. +0.35, as presented in the standard data used in this study. This is true for solid models. However, in the case of placing an opening in any

of the building two windward faces, values of pressure coefficient for the two windward faces in Fluent 5.5 have been found to be different, i.e., +0.28 for the face that has the window, and +0.32 for the other one. This can be noted in Fig. 6, showing contours of static pressure.

7. Conclusion

This paper has compared the use of CFD and Network models for wind-induced ventilation prediction in buildings. It compares the calculated airflow rate using the mathematical Network model and the modelled one using Fluent 5.5 program. Many cases with a variety in building geometries and wind directions have been considered. Results obtained support the use of the proposed CFD code for wind-induced natural ventilation in buildings, as a good agreement has been achieved. This can be recommended as a validation method for studies that have no access to laboratory or full-scale testing facilities. This study has also revealed that the chose of mesh type and size, in addition to the domain size are critical parameters in three-dimensional CFD modelling.

References

[1] CIBSE. Natural ventilation in non-domestic buildings. London: CIBSE; 1997.
[2] Bahadori M, Haghighat F. Passive cooling in hot arid regions in developing countries by employing domed roofs and reducing the temperature of internal spaces. Building and Environment 1985;20(2):103–13.
[3] Chow WK. Wind-induced indoor-air flow in a high-rise building adjacent to a vertical wall. Applied Energy 2004;77:225–34.
[4] Awbi H. Ventilation of buildings, second ed. London: Spon Press; 2003.
[5] Liddament MW. Air infiltration calculation techniques—an application guide. Bracknell. Coventry: The Air Infiltration and Ventilation Centre; 1986.
[6] British Standards Institution. BS 5925: code of practice for ventilation principles and designing for natural ventilation. London: BSI; 1991.
[7] CIBSE., fifth ed. CIBSE guide, vol. A. London: CIBSE; 1988.
[8] Kindamgen J, Krauss G, Depecker P. Effects of roof shapes on wind-induced air motion inside buildings. Building and Environment 1997;32(1):1–11.
[9] Fluent Inc. Gambit 1.3 program help menu; 1988.