DEVELOPING COMPUTATIONAL FLUID DYNAMICS CONDITIONS FOR URBAN NATURAL VENTILAITON STUDY

Jianqiang Li[1]and Ian C. Ward[1]

[1]School of Architecture, the University of Sheffield,
Sheffield, S10 2TN, UK

ABSTRACT

Computational fluid dynamics (CFD) is a promising method to study the urban built environment. However, the pecularities of the urban wind environment are difficult to simulate with a CFD package. The aim of this work is to develop CFD model capable of simulating the urban boundary layer which can then be used to investigate the effects of built form on the pressure coefficient regeme. The model concentrates on two aspcepts: the boundary conditions and domain size. The main methodologies involve developing a CFD boundary lasyer model and testing it against the results of physical modelling in a biundary layer wind tunne. The domain size is dependent on the building area, density and layout. The results of this work are an acceptable cFD model which can then be used to investigate specific problems.

KEYWORDS

CFD; Urban natural ventilation; Boundary conditions; Domain size.

INTRODUCTION

Computational fluid dynamics is a popular method of studying the wind environment and natural ventilation in buildings, due to its advantages on saving time and cost compared to a boundary layer wind tunnel. However, one phenomenon is that most studies of natural ventilation are carried out for an isolated building and do not take into account influences of the surrounding buildings. In reality the surrounding buildings can have significant effects on the wind pressure distribution on the facades of the building under study which are the main driving force for natural ventilation. One difficulty in calculating natural ventilation is obtaining the correct pressure coefficients between inlet and outlet as the local free stream velocity can not be obtained from the local meterological office. The usual method to acquire such coefficients is from a code or standards, but these may not be appropriate as they are too simplified (Khanduri, et al. 1997). However, the current CFD modelling still faces similar problems: The boudnary conditions in urban environments are more complicated than rural or open areas and the turbulence models in CFD can face some problems dealing with the near ground region which is very close to urban built envrionment due to the large roughness height of urban areas (Castro, 1999). Another problem faced by CFD models is the available domain size in urban environments. In wind tunnel studies the downstream and lateral fetch are shorter than the upstream fetch and to simulate these in CFD may lead to some optmisation (Hargaves & Wright, 2006).

METHODOLOGY

Concept

The urban wind profile at the inlet boundary is the most important issue, which determines development the whole domain turbulent flow. There are two types of methods available to describe it in CFD. The most common one is the power law (exponent decided by terrain types) or logarithmic law (friction velocity and roughness length) acquired from wind tunnel experiments or field studies. This method may be useful for the region above the urban roughness sublayer, but they fail to describe the region below the building height (MacDonald, 2000), which is much more important for urban natural ventilation. An alternative approach in CFD is to directly simulate the roughness elements (Mile & Westbury, 2003). This approach may not be acceptable as it results in a significant amount of computational time being devoted to simulating these elements rather than concentrating on the buildings under consideration.

The concept in this paper for the CFD application to the study of urban natural ventilation is derived from the traditional urban boundary layer wind tunnel work carried out by (Lee, 1977) in which the urban wind profile is reproduced by a honeycomb fence, spires and urban roughness elements (figure 1). The boundary layer free velocity (U_G) is 9.65m/s, and the height of model is 0.036m. The Reynolds number (based on the free stream velocity and model height) is about 1.9×10^4, and its roughness Reynolds number is 1,261, so the flow is independent (Snyder and Castro, 2002). Instead of simulating the whole wind tunnel including urban roughness elements and building models, the simulation can be divided into two steps: the first is to acquire the urban wind profile after the infinite length of urban roughness elements, which is then used as input to the CFD domain. The CFD domain size consists of the fully developed flow region and the area in which the building models are situated. The fully developed flow region has been confirmed by many researchers, such as Franke et al. (2004) who recommended that the location of model is less than 5H away from inlet, lateral and top boundary, and 15H in front of outlet boundary. However, the region where the building models are placed can be defined as the region in which the pressure coefficient difference between windward and leeward of the studied building is significantly affected by the surrounding buildings. From this concept, the main tasks are to acquire the inlet boundary conditions and decide the urban natural ventilation neighborhood scale.

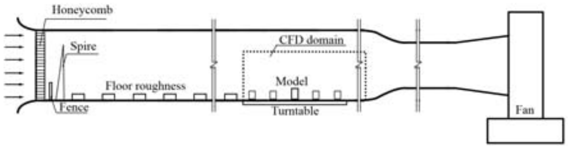

Figure 1 Schematic of CFD domain in wind tunnel

CFD settings

Wind flow can be regarded as incompressible turbulent flow as the density stratification around the buildings is very weak. The numerical solution of the governing equations: the mass conservation (continuity) equations and the momentum conservation equations (Navier-Stokes), is using a finite volume method, and the discretised equations are solved by means of segregated method. Pressure-velocity coupling uses the SIMPLEC algorithm. The solution is based on second order upwind scheme difference. As the wind flow in the urban canopy is totally three-dimension and complicated, the RSM turbulence model is more appropriate to use rather than the two equation turbulence models, such as standard k-ε and its improved turbulence models, because they still have problems in capturing accurate separation and reattachments as well as assuming wind as isotropic flow. Y+ near the ground in the urban canopy is usually less than 30, that means wall function or non-equilibrium wall function may be impossible to apply. So enhanced wall treatment based on two- layer theory is applied. All the calculations were performed using the commercial CFD code FLUENT 6.2 by high performance computing cluster at the University of Sheffield.

NUMERICAL SIMULATION

Modeling urban wind profile

Before the simulations can be carried out it is necessary to build the equilibrium urban wind profile. A wind tunnel achieves this by the extension of the fetch of urban roughness elements, which occupy the main part of the length. This approach can not be used in CFD as computational resources are limited. However, the repeated elements from the wind tunnel can be simplified as one unit (figure 2). The periodic boundary condition was applied into the streamwise direction and lateral boundary. The top boundary is set as symmetrical boundary. The surface stress ($u_\tau = 0.0046U_G{}^2$) is got from the wind tunnel test (Lee, 1977), and the pressure gradient was derived from the following equation:

$$\rho u_\tau{}^2 = \frac{\partial P}{\partial x} \times L_z \qquad (1)$$

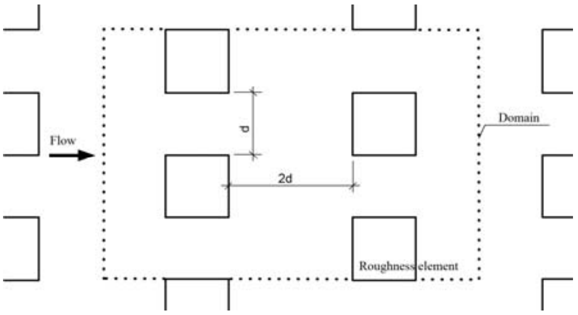

Figure 2 Plan of computational urban roughness elements domain

Three types of grids (table 1) were investigated to study grid sensitivity: Grid G2 is approximately four times that of G1 and less than thirteen times that of G3. The wind profiles are compared with the four types of data: mean velocity ($\overline{V}$)and the mean velocity at the roughness element height ($\overline{V_H}$), the mean turbulence kinetic energy ($\overline{E}$)and the mean turbulence kinetic energy at the roughness element height ($\overline{E_H}$). It can be seen from table 2 that the mean velocity difference among the three types is

Table 1 Details of set-up of grid sensitivity test

GRIDS	DOMAIN GRIDS			GRIDS ON PER ELEMENTS			TOTAL GRIDS
	Lx	Ly	Lz	Nx	Ny	Nz	
G1	60	40	30	10	10	5	70,000
G2	96	64	48	16	16	8	286,720
G3	144	96	72	24	24	12	967,680

less than 5%. The differences in the mean turbulence kinetic energy are negligable. The larger difference between G2 and G3 is less than 5%, although the G3 is 3.375 times of the grid 2. It was therefore decided to use G2 as the one to use.

Table 2 The wind profile difference between grids

GRIDS	DIFFERENCE (%)			
	$\overline{V}$	$\overline{V_H}$	$\overline{E}$	$\overline{E_H}$
G1 VS G2	1.4	2.7	0	2.7
G3 VS G2	-3.4	-1.3	0	-4.2
G1 VS G3	4.9	1.3	0	7.2

The mean veolocity profile and turbulence profile in the urban roughness element area was achived by the area weighted average data. Domain size height is normalized to the building height (H) and mean velocity and TKE are normalized by free stream veloicty (U_G). Figure 3 illustrates the present velocity profile, compared with MacDonald (MD), Kelithn and Roatch (KR) as well as wind tunnel tests (Hussain, 1980). It can be seen that the presenst results shows good agreement with the other methods, especially MD's method. The larger diffrence between the present and MD's method is less than 8%. The data above 2H are very close to the boundary layer wind tunnel results. The differneces below 2H could be due to the fact that the results from the wind tunnel tests are based on the point data while in the current tests the results are area averaged. In genearl, it can be shown that the mean wind profile can be expressed by the urban canopy layer, roughness sublayer, inertial sublayer and outer layer.

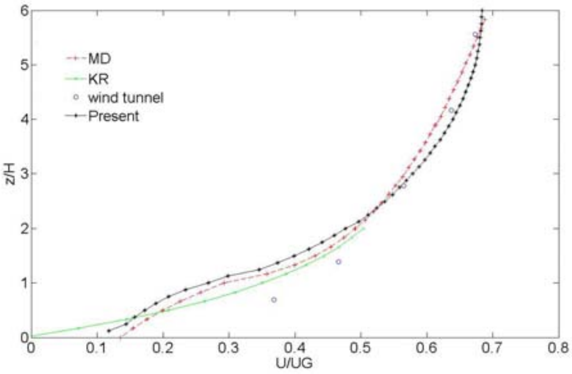

Figure 3 Mean velocity profile in urban roughness elements region

Figure 4 shows TKE profiles obtained from the wind tunnel tests, KR's method and present CFD tests. As wind tunnel test conducted above the building height, not including the region below building height, the valid data are limited. Below the building height, TKE dramatically drops. The maximum TKE is just above the building height, at which surface stress also reaches the maximum. Above the building, TKE profiles are different depending on the method used. The TKE by KR's method is lower than the present study and the wind tunnel tests. However, much research argues that the TKE may looks like a curve (ESDU, 1985, Rao &Nappo, 1998).

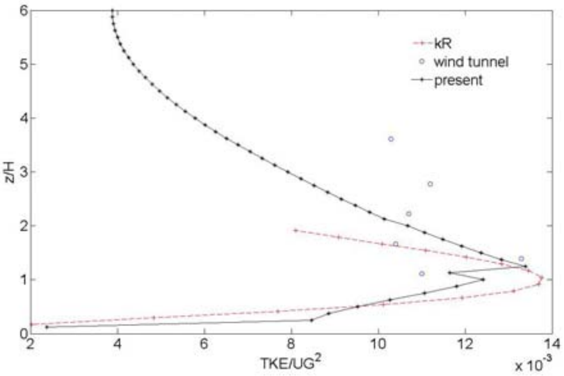

Figure 4 Mean turbulence kinetic energy profile in urban roughness region

Modeling domain size

Computational domain size is dependent on two aspects: the spatial requirement of fully developed turbulence and the working area. The turbulence developed area has been argued and the conclusion according to Cost Action 14 (2004, Franke, et al.) have been used. In the following study, the inlet and lateral boundaries are 6H away from the nearest building and the outlet boundary is 15H behind the last building. The top boundary is 5H away from the building. The neighborhood scale is dependent on the layout, density and the building height. In this study it has been assumed that the building height for all the buildings is the same, which can be argued for an urban area where there are significant numbers of buildings of the same type. The study focuses on the density and layout. According to the three flow regimes: isolated flow, wake interference flow and skimming flow (Lee, 1979), three typical densities: 5%, 10%, and 20% at both of normal and staggered patterns were studied. It can be assumed that the pressure coefficient difference (Cpd) which is the coefficient of wind-induced ventilation force tends to be constant outside neighborhood scale. Pressure coefficient in this study is normalized by the free stream velocity (U_G) instead of free streamwise velocity at building height. The study is divided into two steps: The first step is to investigate the streamwise fetch affects on the pressure coefficient difference between windward and leeward, with the lateral boundary taken as periodical boundary. Ground and building are set as wall boundary with smooth condition. After that, the constant streamwise fetch is achieved and it can be applied into the next step to investigate the effect of the extension the lateral fetch on Cpd.

Normal pattern

Building space (S) and area density (λ) in the normal pattern (figure 5) has the following equation:

$$S = \frac{H}{\sqrt{\lambda}} \tag{2}$$

Taking one array of 12 elements along the wind direction as the object to study the Cpd, (figure 6) describes the effect of streamwise fetch on Cpd. The lowest Cpd can be taken as the constant data which will not be disturbed by the others. In general, the continuous constant data appears along the streamwise fetch. Namely, the fetch can be shorter while considering the effect of fetch in order to save computational resources. If the error compared with the lowest data range is less than 5%, it can be concluded that 10H of fetch can meet it in the upstream fetch. As for the downstream fetch, its effect is lower than the upstream fetch, but the range of 5H should be required. So the total streamwise fetch need 15H in normal pattern layout. In addition, building density has a significant effect on wind-induced ventilation driven force. Cpd of 5% is just one in three of 20% and half of 10%.

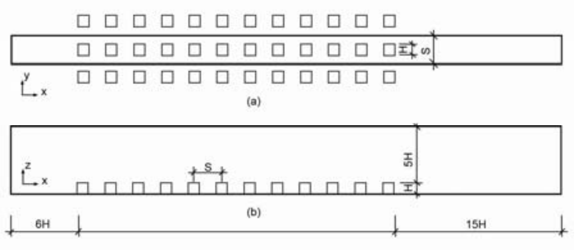

Figure 5 Schematic of normal pattern domain (a) Horizontal section (b) Vertical section

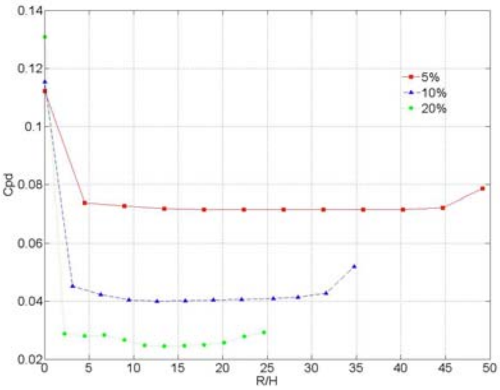

Figure 6 Variation of pressure coefficient difference with fetch - normal pattern

Based on the above streamwise fetch study, table 3 summaries the layouts of three area density in the lateral fetch study. Every lateral fetch is larger than the half of fetch in wind direction. The symmetrical boundary is applied into the lateral boundary, other boundary are same as the above study.

Table 3 Details of layout in lateral fetch study

DENSITY	FETCH IN WIND DIRECTION		FETCH IN LATERAL DIRECTION	
	Elements (No.)	Length (H)	Elements	Length (H)
5%	5	18.77	4	13.91
10%	6	16.8	4	9.98
20%	7	13.44	4	7.22

Figure 7 shows the results of the variation of Cpd with lateral fetch.

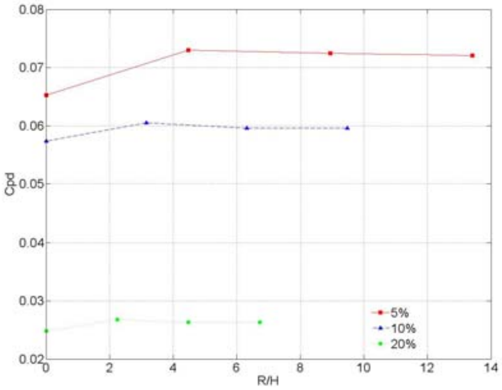

Figure 7 Variation of pressure coefficient difference with lateral fetch - normal pattern

It can be seen that the Cpd's are affected by the nearest elements. In general, 5 times the building height of lateral fetch can meet the requirement of wind-induced ventilation study.

Staggered pattern

A staggered pattern is another typical layout. Assuming that the building height is the same as the building width, the equation of building space and area density can be expressed as:

$$S = \sqrt{\frac{2}{\lambda}} H \tag{3}$$

The first investigation focused on determining the fetch in the wind direction. One unit array can be taken from the repeated units in the lateral direction as objects (figure 8). Lateral boundaries are set as periodic boundary. Other boundary conditions are same as normal pattern.

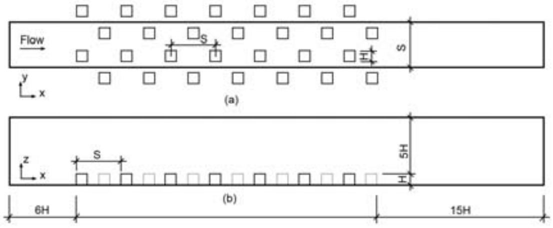

Figure 8 Schematic of staggered pattern domain (a) Horizontal section (b) Vertical section

The effects of fetch on wind pressure coefficient difference between windward and leeward are shown in figure 9. In general, the upstream fetch should be at least 15H, at which spacing the outside building has an effect on Cpd of less than 5% of the lowest value. The downstream fetch requires 5H length. Care should be taken that the upstream fetch in the high density area can significantly affect the wind-induced driven force. The driven force in the 20% area density will be reduced to less than one out of three and one out of five in 5% density.

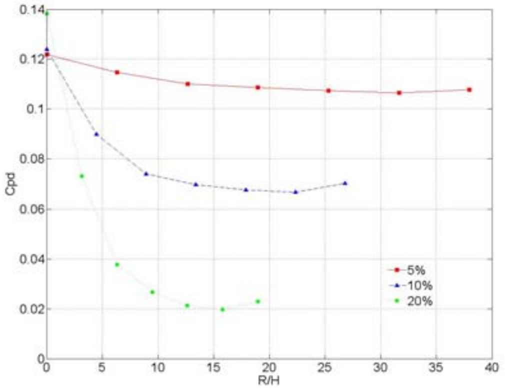

Figure 9 Variation of pressure coefficient difference with fetch – staggered pattern

Figure 10 shows the effect of lateral fetch on the Cpd. In general, the effect of lateral fetch is limited to 10%, compared with the lowest data. The first nearest lateral element plays the vital role of the effect on Cpd. After the first element, other elements along the lateral direction affect it slightly. It can be concluded there that the lateral fetch can be taken the first element.

DISCUSSION

Wind profile in the urban area as mentioned by Richards and Hoxey's method (1993) to define the appropriate boundary condition recommended by COST Action 14 (2004) is not suitable for the urban area. Its assumption of constant turbulence kinetic energy of turbulence contravenes the reality that the turbulence kinetic energy has the maximum value near the building height and declines with the increase of height as the surface stress is highest near the ground. In addition, the wind velocity near the ground cannot be described by the traditional log law or power law. The current test by urban roughness elements study shows that it is necessary to study urban wind profile before simulation of the urban wind. The current test result shows that MacDonald's method is better to describe the mean wind velocity profile. As for the turbulence kinetic energy profile, it appears linear with the height to the building height and at heights above the building height its ; profile is a spline. If we can use turbulence kinetic profile to replace the homogeneous roughness elements, we can more directly focus on the wind profile of the inlet boundary.

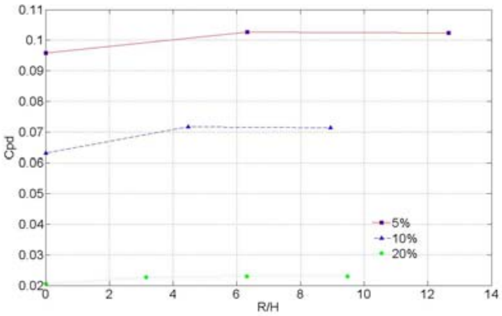

Figure 10 Variation of pressure coefficient difference with lateral fetch - staggered pattern

Computational domain size in the urban area consists of the space for turbulence development and neighborhood scale. The former has been proved by the other researchers. The later is defined for the wind-induced ventilation. Namely, wind pressure coefficient difference between windward and leeward. The results show that the neighborhood scale is dependent on the layout pattern. On the contrary, the area density does not affect the fetch significantly. Table 4 summaries the present results of neighborhood scale against the other researchers.

It generally shows agreement with other researchers. However, the domain of the working area in wind tunnel is decided by the turntable radiometer which is dependent upon the upstream fetch. In CFD approach, it is necessary to distinguish the downstream and lateral fetches in order to efficiently utilize the limited computational resource. The present results show that in the normal pattern, the upstream fetch is 10H, and the downstream and lateral fetch is 5H. For the staggered pattern, the upstream fetch is up to 15H and the downstream and lateral fetch about 6H respectively. Staggered pattern layout required larger domain sizes than the normal pattern. The reason can is due to the larger distance between the neighboring buildings along the wind direction. As the downstream and lateral fetch affect the wind flow, the downstream and lateral fetch of the staggered patterns are similar to the normal pattern.

Table 4 Neighborhood scale for urban natural vententilation

Authors	Method	Longitudinal Fetch			Lateral Fetch
			upstream	downstream	
Cook (1972)	Wind tunnel		5 rows		
Soliman (1976)	Wind tunnel		12H		
Lee, et al (1979)	Wind tunnel		19H	11H	8H
Kiefer & Plate (1998)	Wind tunnel		5H	5H	5H
Franke et al. (2004)	experience		6~10H	6H	6H
Present	CFD	Normal	10H	5H	5H
		Staggered	15H	6H	6H

Wind-induced ventilation potential is significantly affected by the plan layout and area density. Comparing the normal pattern with the staggered pattern at the same area density, 5% (isolated flow regime) and 10% (wake interference flow regime) the staggered pattern has 50% more natural ventilation potential than normal pattern. On the contrary, 20% (skim flow regime) shows that the normal pattern has 20% more natural ventilation potential than the staggered pattern. At the same layout, the lower density has more natural ventilation potential. For instance, the density of 5% is 5 times larger than 20% in staggered pattern, and 3 times in normal pattern.

CONCLUSION

This paper has addressed the issue of the application of CFD in urban natural ventilation, and has proposed a method to optimize CFD boundary conditions as well as domain size for the study of urban natural ventilation. It can be concluded that:

- Urban wind profile for urban natural ventilation is more appropriate to build by urban roughness elements than the common log law or power law profile.

- The computational domain size for urban natural ventilation consists of two parts: one turbulence development, and another for neighborhood scale. The neighborhood scale is dependent upon building layout. Normal pattern needs at least 10 H for upstream fetch and 5H for downstream and lateral fetches. Staggered patterns need at least 15H for upstream fetch and 6H for downstream and lateral fetches.

- Wake interference and isolated flow regimes in staggered pattern and area density have 50% more natural ventilation potential than in normal pattern. The isolated flow regime in the same pattern has at least twice natural ventilation potential than that in the skim flow regimes.

NOMENCLATURE

TKE	Turbulence kinetic energy
H	Building height
Cpd	Pressure coefficient difference between windward and leeward
R	Fetch length
S	Distance between the centre of neighbor buildings
λ	Plan are density
U_G	Boundary layer free stream velocity
Lx,y,z	Domain size in longitude, lateral and vertical direction
P	pressure
U_τ	Surface friction velocity

REFERENCES

Castro, I. P. 2003. “CFD for External Aerodynamics in the Built Environment”. The QNET-CFD Network Newsletter, Vol.2. No.2. 4-7.

Cook, N.J. 1972. “Wind Tunnel Simulation of the Atmospheric Boundary Layer”. Symposium on External Flows”. University of Bristol.

Franke, J. et al. 2004. Recommend Actions on the Use of CFD in Predicting Pedestrian Wind Environment. http://www.costc14.bham.ac.uk.

Hargaves, D. & Wright, N.G. 2006 “On the Use of the k-ε Model in Commercial CFD Software to Model the Neutral Atmospheric Boundary Layer”. Journal of Wind Engineering. [Article in Press]

Hussain, M. & B.E. Lee, B.E. 1980. An Investigation of Wind Forces on Three Dimensional Roughness Elements in A Simulated Atmospheric Boundary Layer: Part 1, BS 55, Department of Building Science, University of Sheffield. 1980.

Kastner-klein, P. & Rotach, M.W. 2004. “Mean Flow and Turbulence Characteristics in An Urban Roughness Sublayer”. Boundary-layer Meteorology. 111. 55-84.

Khanduri, A. C. et al. 1997. “Wind-induced Interference effects on buildings - a review of the state-of- the art”. Engineering Structures, 20**,** 617-630.

Kiefer, H. & Plate, E.J. 1998. "Modelling of Mean and Fluctuating Wind Loads in Built-up Areas." Journal of Wind Engineering and Industrial Aerodynamics 74-76: 619-629.

Lee, B. E. 1977. The Simulation of Atmospheric Boundary Layers in the Sheffield University 1.2 x1.2m Boundary Layer Wind Tunnel. Department of Building Science, Faculty of Architectural Studies, University of Sheffield.

Lee, B.E. et al. 1979. “A Method for the Assesment of the Wind Induced Natural Ventilation Forces Acting on Low Rise Building Arrays2. Department of Building Science. University of Sheffield.

Macdonlad, R.W. 2000. “Modelling the Mean Velocity Profile in the Urban Canopy Layer”. Boundary-layer Meteorology. 97. 25 – 45.

Mile, S. & Westbury,P. 2003. “Practical Tools for Wind Engineering in the Built Environment”. QNET-CFD Network Newsletter. Vol.2. No.2. 11-14.

Riddle, A. et al. 2004. “Comparisons between FLUENT and ADMS for Atmospheric Dispersion Modeling”. Atompheric Environment 33.1029–1038.

Richard P.J. & R. Hoxey, R. 1993. “Appropriate Boundary Layer Conditions for Computational Wind Engineering Models Using the k-e Turbulence Model”. Journal of Wind Engineering and Industrial Aerodynamics, 46, 47. 145 – 153.

Soliman, B.F. 1976. A Study of the Wind Pressure Forces Acting on Groups of Buildings”. PhD Thesis, Department of Building Science, Unviersity of Sheffield.

Snyder, W.H. & Castro, I.P. 2002. “The Critical Reynolds number for Rough-wall Boundary Layers. Journal of Wind Engineering and Industrial Aerodynamics, 90, 41-54.