First International Conference on Sustainability and the Future
FISC2010

BUE-FISC – 0016

ASSESSMENT OF WIND FLOW WITHIN THE BUILT ENVIRONMENT

Islam Abohela / Dr Neveen Hamza / Dr Steven Dudek
School of Architecture, Planning and Landscape, Newcastle University, UK
islam.abohela@ncl.ac.uk

ABSTRACT:
With the growing interest among architects, planners and developers to integrate wind turbines within the built environment to reduce the reliance on grid supplied electricity for buildings to be self sufficient in terms of energy, the importance of assessing wind flow within the built environment becomes more important. This paper reviews existing researches on wind assessment tools. The main tools for assessing wind flow within the built environment are Mathematical Models, In Situ Measurements, Wind Tunnel Tests and Simulation Tools or Computational Fluid Dynamics (CFD) Calculations. The pros and cons of each tool revealed that researchers favour CFD and wind tunnel tests over the other tools. In addition, CFD has proven to be reliable in terms of obtaining data consistent with the results from wind tunnel tests. However, it should be noted that CFD modelling is not a straightforward process and it requires high levels of training and fluid mechanics knowledge for architects to efficiently use the tool. This knowledge would assist architects and planners to confidently specify the right simulation parameters in CFD. One of these parameters is the domain grid which plays an important role in the accuracy of the simulation. Wind flow around a building was simulated with different grid spacing and the simulation proved the discrepancies in the results due to refining the grid. This proves that CFD when used for assessing wind flow within the built environment should be validated using other wind assessment tools. Many studies have proven the reliability of CFD results if the right parameters and conditions were implemented and that makes CFD a good and economic tool for comparing design alternatives. However, this paper argues for the current need to train architects and planners on using CFD software efficiently in order to understand wind flow around and within buildings which would positively affect their design decisions regarding natural ventilation, thermal comfort or integrating wind turbines within the built environment.

Conference Topic: Renewable Energy
Keywords:
Wind, Buildings, Urban, CFD, Wind Tunnel

1. INTRODUCTION:

One of the untapped potentials to reduce harmful carbon gas emissions from buildings is the in situ generated electricity through wind turbines. Manipulating building form to augment and enhance the performance of these wind turbines can lead to decreasing reliance on grid supplied electricity (Abohela, 2009; Taylor, 1998). Although the idea of generating electricity where it is needed using urban wind turbines is attractive for the advantages of reducing the transmission and distribution losses, it is however, a very complicated process due to the complexity of wind flow within the built environment. Wind flow in urban areas is completely different from flow in open fields and offshore. When wind gusts over urban areas the mean wind speed decreases and the flow becomes unpredictable in terms of direction and velocity due to the complexity and the variety of elements forming urban areas such as buildings, concrete sealed grounds, and vegetated gardens (Lei et al., 2006; Syngellakis & Traylor, 2007; Yuen et al., 2004). According to Denoon et al. (2008), turbines work most efficiently in low-turbulence environments, so care needs to be taken in specifying turbine types that will cope with both existing turbulence and likely future changes in turbulence as a result of urban development. This is why it is mandatory to undergo a complete assessment of wind flow within particular urban areas before integrating wind turbines. Campos-Arriaga (2009) confirmed the availability of many tools for assessing wind flow within the built environment which when properly used can lead to informative data which would be implemented in making good design decisions about integrating wind turbines in a building or in an urban area.

2. WIND ASSESSMENT TOOLS:

According to Mertens (2006) and Paterson & Apelt (1989) the research tools used to understand wind flow within the built environment can be divided into: Mathematical Models, In Situ Measurments, Wind Tunnel Tests and Simulation Tools or Computational Fluid Dynamics (CFD) Calculations. All of these tools have specific advantages and drawbacks that define the suitability of the tool for a certain analysis. Existing research in this area (Campos-Arriaga, 2009; He & Song, 1999; Mochida et al., 1997; Murakami et al., 1999) favoured Wind Tunnel Tests and CFD modelling over Mathematical Models and In Situ Measurements. According to Versteeg and Malalasekera, (2007), Mathematical Models are based on the Navier-Stokes equations, which describes the flow behaviour. There are a number of flows that can be approximated with a special case of the Navier-Stoke equation (the EuIer Equation), in which the flow is regarded as homogeneous and inviscid. However, this method is difficult to use, extremely time consuming and requires a thorough knowledge of fluid dynamics. However, there is currently a considerable effort in research to develop other simulation models to solve the turbulence problem. One of these techniques is the use of Large Eddy simulation (LES) techniques in CFD which successfully simulates wind flow at pedestrian level; another model is the Reynolds-averaged Navier-Stokes (RANS) (He & Song, 1999; Lei et al., 2006; Leitl et al., 1997; Malcolm et al., 2007).

As for In situ measurements, Plate (1999) asserted that this tool is probably the most accurate tool for assessing wind flow on a particular site, especially when this assessment is aimed for retrofitting wind turbine into an existing building. In many cases this tool is used as a means of validation for other tools. However, the major drawback of this approach is that this is a very difficult and expensive experiment to carry out even once, yet along, many times. On the other hand, Willemsen & Wisse (2002) asserted that different experiments have proven that in situ measurements are loaded with many errors especially at pedestrian level in the built environment which could reach 20%. This doesn't mean that the other tools are not loaded with errors; both CFD and Wind Tunnel Tests have embedded errors.

3. WIND TUNNEL TESTS:

According to Baskaran & Kashef (1996) and Plate (1999), this tool was originally developed for aeronautic and industrial engineering but was then widely used in testing physical scaled buildings models. However, Blocken & Carmeliet (2004) confirmed that the first wind tunnels did not simulate wind flow correctly because it had wind speed of equal values throughout the cross section of the wind tunnel, which is not the case for the atmospheric boundary layer which is characterized by the variation in mean wind speed with height. Early literature focused on this aspect which resulted in the emergence of wind tunnels which took into consideration the increase in wind speed with height. Campos-Arriaga (2009) acknowledged that the data obtained from these tests are, to a great extent, considered reliable if the wind tunnel used is an atmospheric boundary layer tunnel and the model is accurately constructed with all surrounding elements affecting wind flow. Jones et al., (2004) and Malcolm et al. (2007) also stressed on the importance of including surrounding buildings in the study when introducing a building model for a wind tunnel, because these buildings will have a great effect on wind flow pattern and speed.

On the other hand, Denoon et al. (2008) found it difficult to accurately model the effect of turbulence in a wind tunnel because the wind tunnel is limited by its size, this is why a complete accurate simulation of wind flow is not yet possible, which means that the results obtained from wind tunnel testing will have errors that should be considered. However, in order to increase confidence in the results obtained from Wind Tunnel Tests, Willemsen & Wisse (2002) acknowledged the importance of validating these results using other tools in order to determine the technical and aero dynamical errors. Another drawback of Wind Tunnel Tests which Campos-Arriaga, (2009) highlighted is that these tests are expensive in terms of construction, operation and maintenance. Furthermore, Tominaga and Mochida, (1999) asserted that wind tunnel equipment is not readily available to many planners, designers, and architects. Accordingly, they miss the advantages of implementing wind tunnel tests during the design stage which consequently limits the efficiency of their designs. This problem was overcome by the CFD simulation software which is relatively a new and inexpensive wind assessment tool compared to wind tunnel tests.

3. COMPUTATIONAL FLUID DYNAMICS (CFD) CLACULATIONS:

Asfour and Gadi, (2007) acknowledged that some of the application of CFD in the built environment is in the field of predicting airflow rate, air velocity, air temperature, airflow patterns inside and around buildings and assessing pedestrian wind environment and micro-scale atmospheric environment around human body. Clifford et al. (1997) and Jones & Whittle (1992) predicted that due to the ever-increasing computing power, CFD will be the obvious tool for assessing wind flow within the built environment for the purpose of integrating wind turbines. Even small practices have started using CFD simulation due to their relatively low cost compared to other tools. Currently CFD is used in modelling the potential of introducing natural ventilation in high rise buildings. This could be taken a step further to look into the possibility of enhancing the building form to improve wind power generation. Swiss Re building by Fosters and Partners was extensively modelled to capture natural ventilation in its atriums throughout the building height and assess reducing the wind turbulences on pedestrians. The tapering shape of the bottom part of the building is a direct respond to the CFD simulation results to provide a comfortable pedestrian wind environment (Figure.1) (Kitson, M., & Moran, H., 2006).

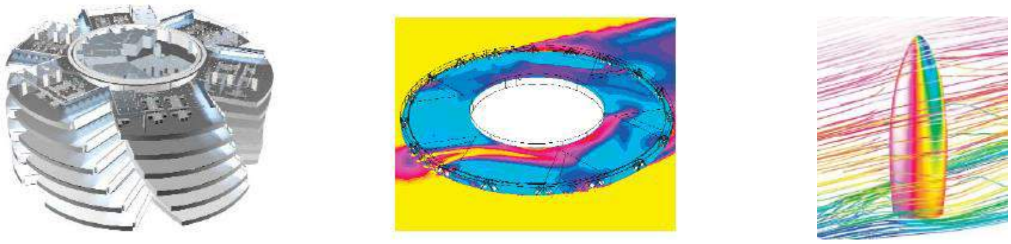

Fig.1 Swiss Re Building
A CAD model of several floors in the Swiss Re building (left), a CFD simulation showing airflow through one of the floors (middle) and a CFD simulation of air flow around the whole building (right)

According to Versteeg and Malalasekera, (2007) and Asfour & Gadi, (2007) CFD is based on solving the fundamental governing equations of fluid dynamics that describe the exact behaviour of a Newtonian fluid, including the effects of turbulence. Assuming that the flow is incompressible, the following equations can be used to describe the fluid flow,

- Navier-Stokes Equations: (conservation of momentum)

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial u_i}{\partial x_j}\right) \qquad \text{Eq. 1}$$

- The continuity equation: (conservation of mass)

$$\frac{\partial u_i}{\partial x_j} = 0 \qquad \text{Eq. 2}$$

Where u is the velocity in the streamwise direction, p is the pressure, ρ is the fluid density and υ is the kinematic viscosity of the flow.

Many researches (He & Song, 1999; Mochida et al., 1997; Murakami et al., 1999; Stathopoulos, 2006) agreed about the main advantage of CFD being its powerful visualisation tools. Augenbroe (2004) and Jones et al. (2004) acknowledged that the main advantage of CFD simulation which encourages architects and engineers to broadly use it is the speeding up that happens in the design process when using the CFD tool which also gives the designer the possibility of making modifications and comparing different scenarios quickly. Moreover, simulation tools can provide with a better understanding of the consequences of design decisions, which increases the effectiveness of the design process as a whole. In addition, Stathopoulos (2006) confirmed that CFD modelling is less expensive than full scale measurements and wind tunnel experiments. Moreover, Versteeg and Malalasekera, (2007) asserted that there are several advantages of CFD such as the substantial reduction of lead times and costs of new designs and comparing alternatives, the ability to study unreachable conditions and the ability to produce practically unlimited level of detail of results.

However, Denoon et al. (2008) claimed that CFD simulation is so weak in the field of assessing wind flow in very dense urban environments; this is attributed to the insufficient computational power available to accurately model the effects of turbulence in the built environment. In light of this and since wind flow in urban areas is unpredictable, CFD is considered a relevant tool for comparison between different designs and not a relevant tool for accurately assessing wind speeds in urban areas for energy yield calculations. A combination of wind tunnel tests and CFD simulation would be relevant for such kind of investigations. Campos-Arriaga, (2009) and Syngellakis & Traylor (2007) acknowledged that in order to produce reliable simulations, sensible knowledge of fluid dynamics is required; proficiency in the use of the specific and thoroughly validated CFD code is needed along with related computational skills. Sørensen and Nielsen (2003) also noted that other constrains lie in the choice of the turbulence model and the grid configuration; nevertheless, it is important to realize that even if the governing equations of the given problem are solved accurately, the result may be wrong if the governing equations and boundary conditions do not model the intended physics, like using the wrong turbulence model or a coarse grid as will be demonstrated later, for example CFD is reliable when assessing wind flow upstream of an obstacle but downstream there will be a degree of error using certain turbulence models such as the k-ε turbulence model which is a model based on solving equations representing turbulent properties of the flow by determining the scale of the turbulence (ε) and the energy in the turbulence (k), as for the flow predicted around the obstacle, it will be broadly correct. Existing literature implementing CFD as a tool for assessing wind flow inside and around buildings specified different conditions regarding boundary layers, turbulence models and grid types. The following table (Table.1) summarises some of those conditions in relevant studies.

Table.1 Conditions used for studies of wind flow around buildings using CFD

Author	**Mesh Type**	**Turbulence Model**	**Boundary Conditions**					**CFD Code**
			Inlet	**Outlet**	**Top**	**Bottom**	**Sides**	
(Asfour and Gadi, 2007)	Hex-map	Standard k-ε	Velocity inlet	Outflow	Wall	Wall	Wall	Fluent 5.5
(Blocken et al., 2007)	Structured hexahedral Unstructured wedged Unstructured tetrahedral	Realizable k-ε	Incident power law velocity, turbulence and ε profiles form experimental data	Zero static pressure	Slip wall	Wall with standard wall functions	Sl p wall	Fluent 6.1.22
(Campos-Arriaga, 2009)	Tetrahedral	Realizable k-ε	Velocity profile specified using UDF	Outflow	Symmetry	Wall with standard wall functions	Symmetry	Fluent 6.1.22
(Hargreaves and Wright, 2007)	No-uniform structured grid	Standard k-ε	Velocity, k and ε profiles specified from experimental data	Pressure outlet	Constant sheer stress	Symmetry	Symmetry	Fluent 6.1.22 and CFX
(Hu and Wang, 2005)	Uniform structured grid	Standard k-ε	Incident power law velocity, turbulence and ε profiles	Gauge pressure = Zero	Non-slip wall	Wall with standard wall functions	Symmetry	PHOENICS

			form experimental data					
(Huang et al., 2007)	Combined grid (Unstructured and Structured)	LES and standard k-ε	Velocity inlet	Outflow	Slip condition	Ground	Slip condition	Fluent
(Kim et al., 2009)	Tetrahedral	Standard k-ε	Velocity inlet	Free slip	Wall	Wall	2D	Star-CD
(Mertens, 2006)	Not specified	Realizable k-ε RSN	Velocity, k and ε profiles specified from logarithmic profile equations	Velocity, k and ε profiles specified from logarithmic profile equations	Velocity, k and ε profiles specified from logarithmic profile equations	Wall with standard wall functions	Not specified	Fluent 6.1.18
(N. Meroney et al., 1999)	Combined grid (Unstructured and Structured)	Standard k-ε RNG k-ε RSM	Incident power law velocity, turbulence and ε profiles form experimental data	Outflow	Symmetry	Wall	Symmetry	Fluent 4.4.8
(Xiaomin et al., 2006)	No-uniform structured grid	Standard k-ε	Inlet conditions	Free boundary conditions V=0	Free boundary conditions	Wall	Symmetry	PHOENICS

Chen & Zhai (2004) concluded about CFD modelling that it is not a straightforward process; there is evidence that even with a specific training in fluid dynamics, obtaining accurate results with CFD simulations is not a simple task and it would better be used as a tool for comparing alternatives. One of the main variables affecting the accuracy of the simulation results is the implemented grid in terms of type and spacing. Sørensen and Nielsen (2003) acknowledged that the accuracy of the numerical solution will usually improve with an increased number of grid points, especially if the increase is made in spatial regions with complex geometries. For this reason Asfour and Gadi (2007) asserted that the creation of the mesh (or grid) is one of the most important issues to consider for a successful CFD simulation. A common practice here is to create a hierarchy in mesh size to be fine around the building and coarse away from 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. Versteeg and Malalasekera (2007) acknowledged that the only way to eliminate inaccuracy due to coarseness of a grid is to perform a grid dependence study, which is a procedure of successive refinement of an initially coarse grid until certain key results do not change. Then the simulation is grid independent. Although this process is time consuming but a systematic search for grid-independent results forms an essential part of all high-quality CFD studies.

To demonstrate the importance of grid size for the accuracy of a CFD simulation, FLUENT 12.1 was used in simulating air flow around a building of dimensions 3 x 3 x 3 m with different grid sizes (Figure.2). It was assumed that wind flow within the built environment is turbulent which will affect the simulation model to be chosen. Assuming that air velocity is 5m/s, air density is 1.225 kg/m3, air viscosity is 1.7894e-5 kg/ms and domain length is 21 m. This make the Reynolds number (Eq. 3) equals to 7188163 which means that the turbulent flow is clearly expected.

$$R_e = \rho v l / \mu \qquad \text{Eq. 3}$$

R_e is Reynolds number, ρ is the fluid density, v is the mean flow velocity, l is the length of the domain length and μ is the fluid's dynamic viscosity.

GAMBIT (a pre-processor for FLUENT) is used to create the geometry and the computational grid. The geometry of the problem is constructed in 2D structured grid of 0.1 m (fine grid), 0.5 m (Medium grid) and 1.0 m (coarse grid) interval sizes spacing in both x and y directions with domain size 39*21 m. In order to examine the effect of grid size on the results obtained, wind velocity around the three cases was graphically represented using velocity contours (Figure.3). In addition, velocity magnitude was plotted along three vertical line/rakes at points where x= 18 (l1), 19.5 (l2) and 21 (l3) respectively (Figure.2). The standard procedure to construct the geometry and the grid is to first specify the vertices, then the edges, then the faces, afterwards meshing the edges of the geometry then the faces. In GAMBIT, two zone type specifications were defined; the first is the boundary type specification which defines the physical and operational characteristics of the model and the second type is continuum type which defines the volume entity being solid or fluid. As shown in Figure.2, the left boundary was defined as velocity inlet, the right boundary as pressure outlet and the rest boundaries as stationary walls. As for the continuum it was defined as fluid.

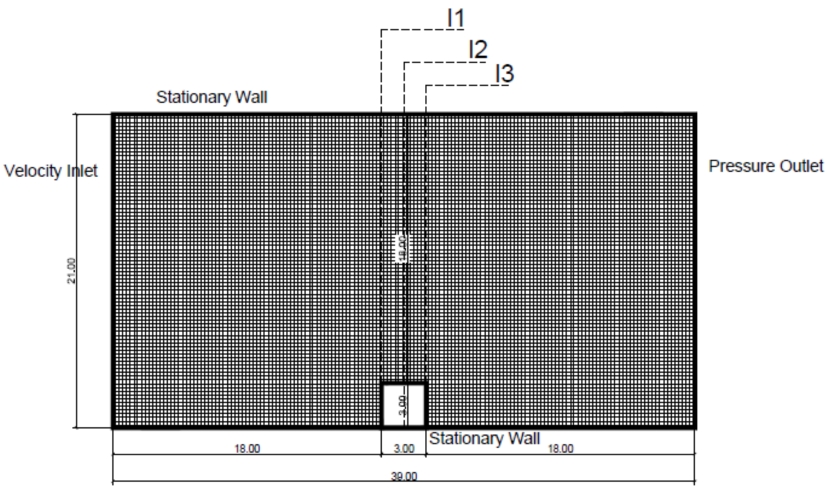

Fig.2 Domain geometry and specifications

After exporting a 2D mesh file from GAMBIT, a 2D double precision solver was chosen in FLUENT. The case file was checked for errors and the solver default parameters were used (Pressure based type, absolute velocity formulation, steady time and planar 2D space). As the temperature distribution is not the point of investigation here, the energy model was kept unchecked. As for the flow turbulence model; the Realizable k-ε turbulence model was chosen. And the material by default air with the default density and viscosity (1.225 kg/m3 and 1.7894e-5 kg/ms respectively). For the operating conditions, the operating pressure was

1atm (101325 Pascal). As for the boundary conditions, it was predefined in GAMBIT so the only parameter to be defined is the velocity inlet value which is 5.00 m/s. The second order upwind was chosen for the momentum, turbulent kinetic energy and turbulent dissipation rate equations. The chosen convergence criterion for all equations was set to 1e-6 and the number of iterations was set to 100. And the contour of velocity magnitude was plotted for the three cases (Figure.3, 4 & 5).

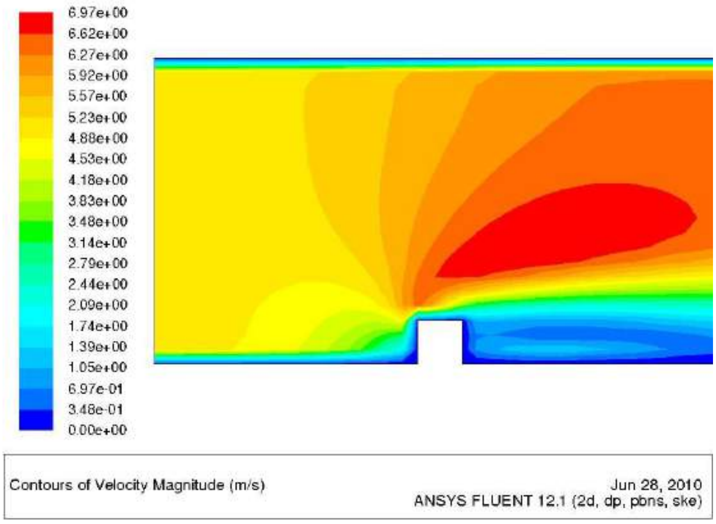

Fig.3 Velocity Contours of velocity magnitudes for coarse mesh

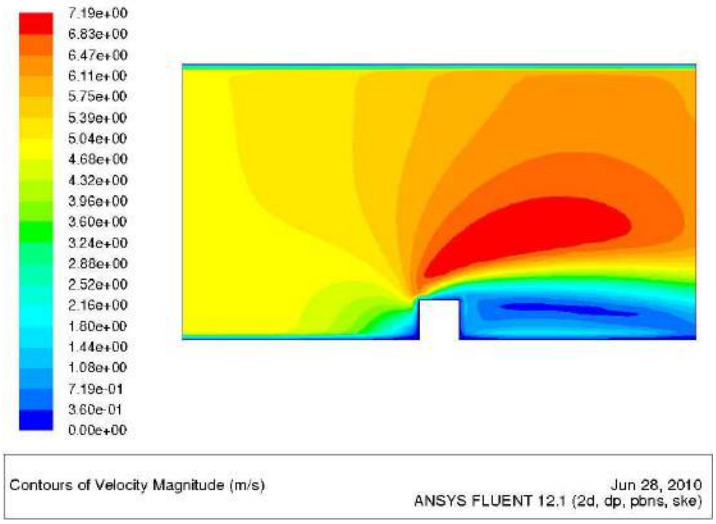

Fig.4 Velocity Contours of velocity magnitudes for medium mesh

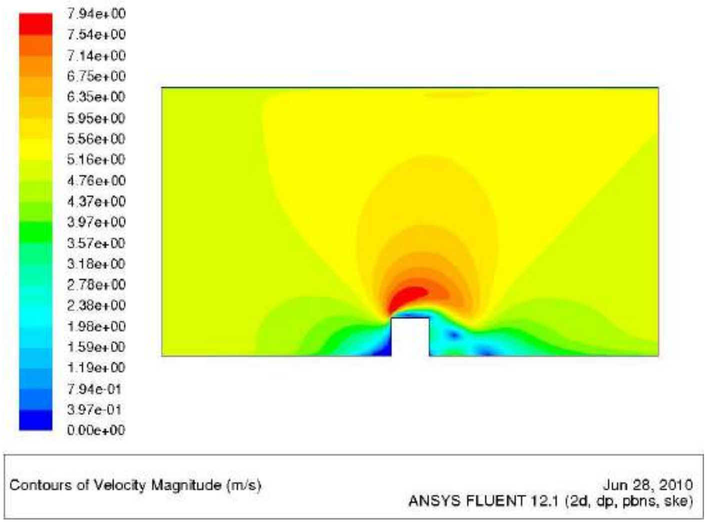

Fig.5 Velocity Contours of velocity magnitudes for coarse fine mesh

The contours of velocity magnitude revealed the different distribution of velocity in the domain for different grid sizes. It was noticed that the areas of highest discrepancies were above and on the downstream of the building. The coarse and medium grid failed to capture the two vortices which were captured by the fine grid downstream the building. In addition, the area of velocity augmentation above the building was over estimated by the coarse and medium grid. By plotting velocity magnitude along the three lines, the differences in velocities were recorded. However, these differences varied along the three lines; along line l1 (Figure.6) the highest difference was recorded at 3.9m from the ground level, the coarse and medium grid recorded velocity magnitude of 6.3 m/s while the fine grid recorded velocity magnitude of 7.9 m/s. Along line l2 (Figure.7) the highest difference was recorded at 4.9m from the ground level, the coarse grid recorded velocity magnitude of 6.2 m/s, the medium grid recorded velocity magnitude of 6.9 m/s while the fine grid recorded velocity magnitude of 7.9 m/s. Along line l3 (Figure.8) the highest difference was recorded at 1.7m from the ground level, the coarse grid recorded velocity magnitude of 0.7 m/s, the medium grid recorded velocity magnitude of 1.2 m/s while the fine grid recorded velocity magnitude of 3.8 m/s. Although the difference between the highest and lowest recorded velocities along the three lines varied between 1.6 m/s and 3.1 m/s which can be considered relatively small values, these values have high significance when assessment of wind flow is carried out for the purpose of integrating wind turbines within the built environment because the energy yield of a wind turbine is directly proportional to cube the wind speed according to the following equation:

$$P_{turb}=C_p 1/2\ \rho A V^3 \quad \text{Eq. 4}$$

Where P_{turb} is the power from the turbine, C_p is the coefficient of performance, ρ is the air density, A is the swept area of the blades and V is the free wind velocity. This means that if the wind speed doubles the energy yield will increase eight times (Stankovic et al., 2009).

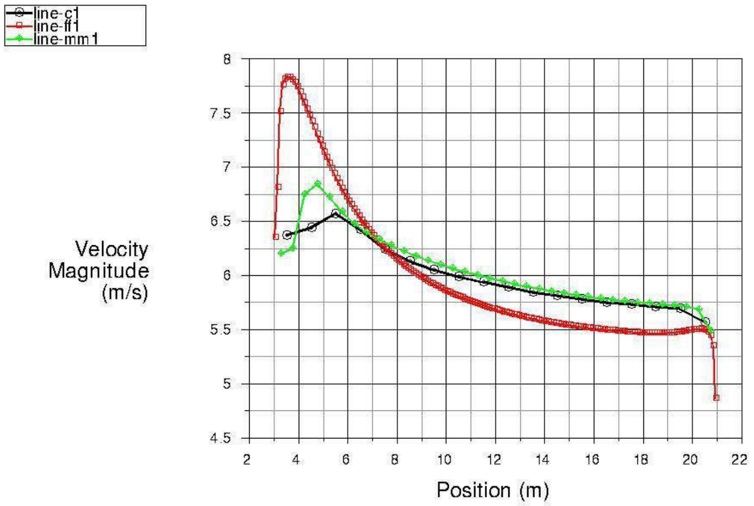

Fig.6 Velocity magnitude plot along line l1 for coarse (c1), medium (mm1) and fine (ff1) meshes

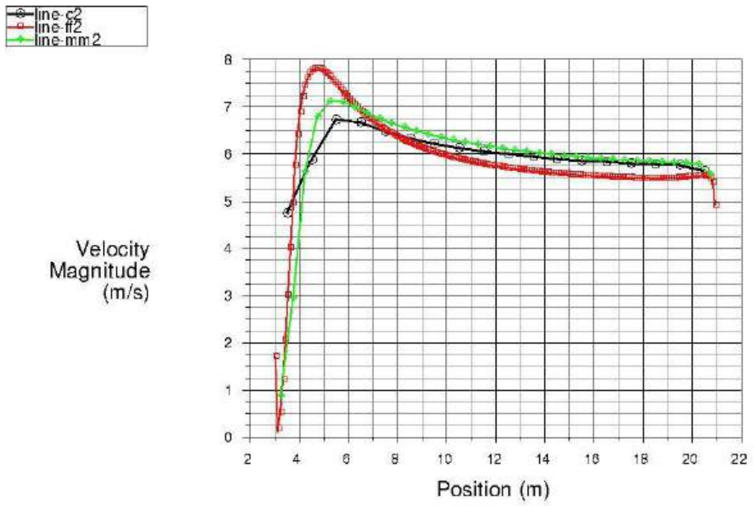

Fig.7 Velocity magnitude plot along line l2 for coarse (c1), medium (mm1) and fine (ff1) meshes

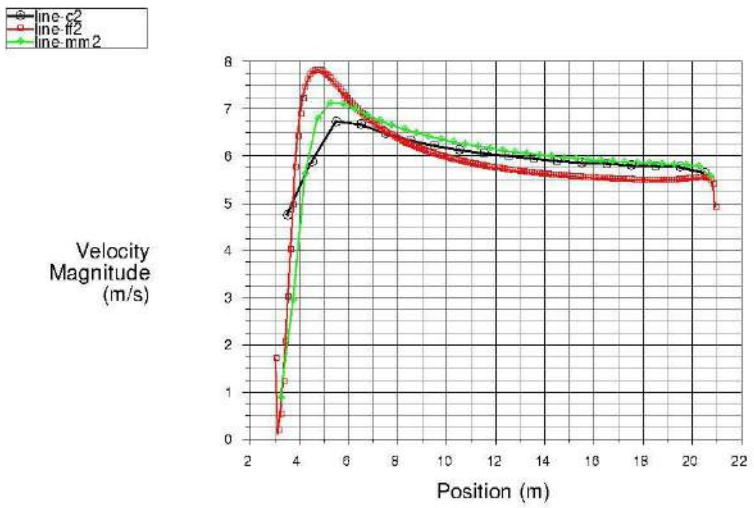

Fig.8 Velocity magnitude plot along line l3 for coarse (c1), medium (mm1) and fine (ff1) meshes

4. Discussion

Although, engineers, planners and architects favour both CFD calculations and wind tunnel tests over in situ measurements and mathematical models, Blocken & Carmeliet (2004) and Chen (2004) argued that CFD simulation can provide an alternative for wind tunnel studies because CFD is less time consuming and less expensive than wind tunnel tests and it is easy to visualise the detailed wind flow within the domain of study but in order to use this tool in confidence the model should be validated using other tools. However, when comparing the results obtained from wind tunnel tests and CFD calculations, Jiang et al. (2008) asserted that CFD simulations agree well with wind tunnel test in the flow field and wind pressure distribution around buildings but the differences between the results obtained from the two tools are more apparent at ground levels. Campos-Arriaga (2009) attributed this to the treatment used at the near wall region (roughness and mesh specifications at pedestrian level) in the CFD simulation tool. The near wall treatment significantly influences the accuracy of numerical solutions, because it is in that region where the solution variables have large gradients, and the momentum and other scalar transports occur most vigorously. Therefore, representing the flow in these regions accurately will lead to good turbulence simulation and accordingly good results. Jones et al. (2004) also noticed some discrepancies in results when assessing simple and complex form. However, there is agreement in general flow trends, which means that the problem is in the wind environment simulated in both tools. In practical application, these differences could lead to different design decisions which mean that further work is required for identifying these problems in both detailed wind-tunnel measurements and CFD turbulence simulation.

Since all the tools available for assessing wind flow within the built environment are relatively expensive and since most accurate results could be obtained by in situ measurements which is the most expensive and time consuming tool, therefore CFD simulation and wind tunnel testing need to be developed to improve the estimation of urban wind speeds without having to rely on in situ measurements. The following table (Table.2) concludes the observations about wind assessment tools in the built environment in terms of accuracy, usage as a visualisation tool, preference of usage for existing and future planed developments, cost, required time for assessment and availability to architects.

Table.2 Comparison between wind assessment tools

	Tools arranged in descending order
High Accuracy	In Situ Measurements – Wind Tunnel – CFD – Mathematical Models
High Visualization	CFD - Wind Tunnel - In Situ Measurements - Mathematical Models
Assessing wind flow in existing urban areas	In Situ Measurements – CFD – Wind Tunnel – Mathematical Models
Assessing wind flow for future planned urban areas	CFD – Wind Tunnel – In Situ Measurements – Mathematical Models

Lowest cost	Mathematical Models - CFD – Wind Tunnel – In Situ Measurements
Less time consumed	CFD – Wind Tunnel – Mathematical Models - In Situ Measurements
Availability to architects	CFD – Wind Tunnel – In Situ Measurements – Mathematical Models

Since the idea of integrating wind turbines within the built environment is gaining more acceptance among architect and planners and in order to ensure the success and feasibility of such approach, a complete assessment of wind flow characteristics in the proposed site should be carried out. Of all the tools used for assessing wind flow in the built environment, architects favour CFD simulation over other available tools because of its potentials in comparing design alternatives, its high visualisation representation and its comparative ease of use when compared with other assessment tools. On the other hand, it should be noted that CFD simulation is not an accurate tool and should be validated by other available tools which is the same case for all wind assessment tools because they are all embedded with errors. Calculating these errors and taking account of them is very important when assessing the feasibility of integrating wind turbines into buildings. In addition, it should be noted that architects should receive proper training in the field of assessing wind flow within the built environment using the available tools especially CFD simulation. Architects and planners should get involved more in using wind assessment tools to take advantage of the potentials of understanding wind flow around buildings whether through implementing the knowledge in providing buildings with natural ventilation, thermal comfort or integrating wind turbines within the built environment.

Acknowledgements

This work is supported by the School of Architecture, Planning and Landscape in Newcastle University, UK.

References

1. Abohela, I. (2009). **The Integration of Wind Turbines in Architectural Design.** Third Ain Shams University International Conference on Environmental Engineering. Cairo, pp. 01-14.
2. Asfour, O. S. and Gadi, M. B. (2007). **A comparison between CFD and Network models for predicting wind-driven ventilation in buildings.** *Building and Environment,* 42, (12), pp. 4079-4085.
3. Augenbroe, G. (2004). **Trends in building simulation**, in Malkawi, A. M. and Augenbroe, G.(eds) *Advanced Building Simulation.*Oxfordshire: Spon Press.
4. Baskaran, A. and Kashef, A. (1996). **Investigation of air flow around buildings using computational fluid dynamics techniques.** *Engineering Structures,* 18, (11), pp. 861-873.
5. Blocken, B. and Carmeliet, J. (2004). **Pedestrian Wind Environment around Buildings: Literature Review and Practical Examples.** *Journal of Thermal Envelope and Building Science,* 28, (2), pp. 107-159.
6. Blocken, B., Carmeliet, J. and Stathopoulos, T. (2007). **CFD evaluation of wind speed conditions in passages between parallel buildings--effect of wall-function roughness modifications for the**

atmospheric boundary layer flow. *Journal of Wind Engineering and Industrial Aerodynamics*, 95, (9-11), pp. 941-962.

7. Campos-Arriaga, L. (2009). **Wind energy in the built environment: a design analysis using CFD and wind tunnel modelling approach.** PhD thesis. University of Nottingham.
8. Chen, Q. and Zhai, Z. (2004). **The use of Computational Fluid Dynamics Tools for Indoor Environmental design**, in Malkawi, A. M. and Augenbroe, G.(eds) *Advanced Building Simulation.* Oxfordshire: Spon Press.
9. Clifford, M. J., Everitt, P. J., Clarke, R. and Riffat, S. B. (1997). **Using computational fluid dynamics as a design tool for naturally ventilated buildings.** *Building and Environment*, 32, (4), pp. 305-312.
10. Denoon, R., Cochran, B., Banks, D. and Wood, G. (2008). **Harvesting Wind Power from Tall Buildings.** Available at: http://www.ctbuh.org/Portals/0/Repository/T8_DenoonWood.357b6d8b-96e3-4988-b1d6-240cbafe12e3.pdf (Accessed: 20-01-2010).
11. Hargreaves, D. M. and Wright, N. G. (2007). **On the use of the k-[epsilon] model in commercial CFD software to model the neutral atmospheric boundary layer.** *Journal of Wind Engineering and Industrial Aerodynamics*, 95, (5), pp. 355-369.
12. He, J. and Song, C. C. S. (1999). **Evaluation of pedestrian winds in urban area by numerical approach.** *Journal of Wind Engineering and Industrial Aerodynamics*, 81, (1-3), pp. 295-309.
13. Hu, C.-H. and Wang, F. (2005). **Using a CFD approach for the study of street-level winds in a built-up area.** *Building and Environment*, 40, (5), pp. 617-631.
14. Huang, S., Li, Q. S. and Xu, S. (2007). **Numerical evaluation of wind effects on a tall steel building by CFD.** *Journal of Constructional Steel Research*, 63, (5), pp. 612-627.
15. Jiang, Y., Liu, H., Zhang, B., Zhu, F., Liang, B. and Sang, J. (2008). **Wind flow and wind loads on the surface of a towershaped building: Numerical simulations and wind tunnel experiment.** *Science in China Series D: Earth Sciences*, 51, (1), pp. 103-113.
16. Jones, P. J., Alexander, D. and Burnett, J. (2004). **Pedestrian Wind Environment Around High-Rise Residential Buildings in Hong Kong.** *Indoor and Built Environment*, 13, (4), pp. 259-269.
17. Jones, P. J. and Whittle, G. E. (1992). **Computational fluid dynamics for building air flow prediction--current status and capabilities.** *Building and Environment*, 27, (3), pp. 321-338.
18. Kim, T., Kim, K. and Kim, B. S. (2009). **A wind tunnel experiment and CFD analysis on airflow performance of enclosed-arcade markets in Korea.** *Building and Environment*, 45, (5), pp. 1329-1338.
19. Kitson, M. and Moran, H. (2006). **Taking the Built Environment to New Heights.** Available at: http://www.fluent.com/about/news/newsletters/06v15i2/a5.pdf (Accessed: 16 February 2010).
20. Lei, L., Fei, H., Xue-Ling, C., Jin-Hua, J. and Xiao-Guang, M. (2006). **Numerical simulation of the flow within and over an intersection model with Reynolds-averaged Navier–Stokes method.** *Chinese Physics*, 15, (1), pp. 149-155.
21. Leitl, B. M., Kastner-Klein, P., Rau, M. and Meroney, R. N. (1997). **Concentration and flow distributions in the vicinity of U-shaped buildings: Wind-tunnel and computational data.** *Journal of Wind Engineering and Industrial Aerodynamics*, 67-68, pp. 745-755.
22. Malcolm, A. H., John, D. W. and Simon, J. W. (2007). **Estimating the potential yield of small building-mounted wind turbines.** *Wind Energy*, 10, (3), pp. 271-287.
23. Mertens, S. (2006). *Wind* **energy in the built environment: concentrator effects of buildings.** Essex: Multi-Science.
24. Mochida, A., Murakami, S., Ojima, T., Kim, S., Ooka, R. and Sugiyama, H. (1997). **CFD analysis of mesoscale climate in the Greater Tokyo area.** *Journal of Wind Engineering and Industrial Aerodynamics*, 67-68, pp. 459-477.
25. Murakami, S., Zeng, J. and Hayashi, T. (1999). **CFD analysis of wind environment around a human body.** *Journal of Wind Engineering and Industrial Aerodynamics*, 83, (1-3), pp. 393-408.
26. N. Meroney, R., Leitl, B. M., Rafailidis, S. and Schatzmann, M. (1999). **Wind-tunnel and numerical modeling of flow and dispersion about several building shapes.** *Journal of Wind Engineering and Industrial Aerodynamics*, 81, (1-3), pp. 333-345.
27. Paterson, D. A. and Apelt, C. J. (1989). **Simulation of wind flow around three-dimensional buildings.** *Building and Environment*, 24, (1), pp. 39-50.
28. Plate, E. J. (1999). **Methods of investigating urban wind fields--physical models.** *Atmospheric Environment*, 33, (24-25), pp. 3981-3989.
29. Sørensen, D. N. and Nielsen, P. V. (2003). **Quality control of computational fluid dynamics in indoor environments.** *Indoor Air*, 13, (1), pp. 2-17.
30. Stankovic, S., Campbell, N. and Harries, A. (2009). **Urban wind energy.** London: Earthscan.
31. Stathopoulos, T. (2006). **Pedestrian level winds and outdoor human comfort.** *Journal of Wind Engineering and Industrial Aerodynamics*, 94, (11), pp. 769-780.

32. Syngellakis, K. and Traylor, H. (2007). **Urban Wind Resource Assessment in the UK, An introduction to wind resource assessment in the urban environment.** Chineham: European Commission's Intelligent Energy Europe Agency.
33. Taylor, D. (1998). **Using buildings to harvest wind energy.** Building Research & Information, 26, (3), pp. 199-202.
34. Tominaga, Y. and Mochida, A. (1999). **CFD prediction of flowfield and snowdrift around a building complex in a snowy region.** *Journal of Wind Engineering and Industrial Aerodynamics,* 81, (1-3), pp. 273-282.
35. Versteeg, H. K. and Malalasekera, W. (2007). **An introduction to computational fluid dynamics : the finite volume method.** Harlow: Prentice Hall.
36. Willemsen, E. and Wisse, J. A. (2002**). Accuracy of assessment of wind speed in the built environment.** *Journal of Wind Engineering and Industrial Aerodynamics,* 90, (10), pp. 1183-1190.
37. Xiaomin, X., Zhen, H. and Jiasong, W. (2006). **The impact of urban street layout on local atmospheric environment.** *Building and Environment*, 41, (10), pp. 1352-1363.
38. Yuen, C., Zanchetta, M. and Battle, G. (2004). **Performance of a Building Integrated Wind Farm.** *The 21st Conference on Passive and Low Energy Architecture (Plea).* The Netherlands.