# Wind Power Production in the Urban Environment

S.J. Kooiman, S.W. Tullis*

*Department of Mechanical Engineering, McMaster University*
*Hamilton, ON, L8S 4L7, Canada*

Email: *kooimasj@mcmaster.ca*

## ABSTRACT

The purpose of this study is to resolve accurate, local flow details around buildings in an urban environment using commercial computational fluid dynamic (CFD) software. At present, few recommendations have been published regarding the appropriate turbulence models to be used for modelling these flow conditions. Due to the uncertain reliability of the k-$\varepsilon$ models ability to successfully simulate flow over buildings, the current objective is to assess how applicable the k-$\varepsilon$ turbulence model is in resolving this type of flow. Flow around buildings in the atmospheric boundary layer are characterized as having points of separation, reattachment, stagnation and various types of vortices as well as being anisotropic and transient in nature. The complexity of this problem indicates that the turbulence model must be capable of handling many issues to correctly model this flow. In order to justify an applicable turbulence model, these models must be tested under relatively simple conditions and compared with well-documented, large-scale tests to ensure accuracy. The standard k-$\varepsilon$ model was investigated in these conditions because of its relative robustness and past ability to capture the major flow details in this type of flow. Increasing the number of cells in the domain had various effects on the solution. An accurate solution was obtained for the initial mesh and then subsequently the solution diverged and then converged when refining the mesh. Very fine grids were required for a two-dimensional building to achieve accurate results, but the solution was not verified as being grid independent. Even if the solution is grid independent, implementation of the k-$\varepsilon$ model in three-dimensional complex flows would not be practical due to the substantial increase in computational power required. Therefore alternative turbulence models are now being investigated for modelling flow over buildings in the atmospheric boundary layer.

## 1. INTRODUCTION

With the growing demand for renewable clean power production, implementing Vertical Axis Wind Turbines (VAWTs) in urban settings is currently being assessed. Initially, the placement of a wind turbine on top of buildings may seem less than ideal considering the complex flow structure, high directional variability, large skew angles and increased turbulence intensity, all of which are typical deterrents to operating wind turbines. However, with the aerodynamic performance advantages of omni-directionality characteristics, increased performance in the VAWT in skewed flows and the heightened wind speeds around buildings, the application for these turbines may be more advantageous than originally considered. In addition, there are practical benefits of power generation and consumption in the same location. In order to optimize the placement of the wind turbines on top of buildings and minimize negative effects, details of the flow around buildings must be resolved. CFD offers a method of obtaining estimations of the flow fields around various shaped buildings with various fetch characteristics (i.e. the area in front of the building), which are currently not available.

## 2. FLOW STRUCTURE

Flow over a building is multifaceted even in the most simplest of cases when steady state flow is perpendicular to the building's face and no other interferences are present. As the flow approaches the building, a high pressure region is formed on the front face of the structure. This high pressure region directs the incoming air around and above the building. This creates a standing vortex in front of the building as well as a stagnation location

* Tel.: +1-905-525-9140 ext. 27692; Fax: +1-905-525-7944; E-mail: stullis@mcmaster.ca

approximately three quarters up the face of the building. Separation of the flow occurs around the sides and top of the building's edges creating low pressure recirculation zones that are referred to as the separation bubbles. The size of the separation bubble is dependent on the surface roughness, wind velocity and turbulence intensities. The flow over and beside the building will reattach, provided the building is long enough, and will separate off of the back edges of the structure. The normal flow characteristics are schematically shown below in figure 1.

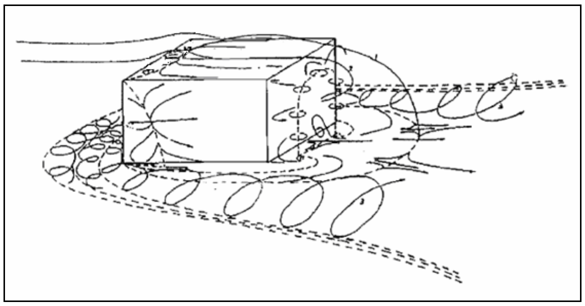

Figure 1: General flow visualization with flow normal to building face. [1]

When the flow is not parallel to the face of the building, cornering vortices form along the edges. The size and angle of the vortex core is dependent on the angle that the flow makes with the building. These are considered delta-wing vortices which develop due to the pressure difference between the high pressure sides and the low pressure top of the building.

The implications of mounting small-scale wind turbines on building rooftops in the urban environment have only briefly been studied. Mertens [2] investigated the wind potential over one specific building configuration and considered two fetch roughness lengths. However, the results of this study were not validated with measured data.

There have been few full-scale tests measuring wind velocities over buildings. Detailed full-scale tests have been performed on the Wind Engineering Research Field Laboratory (WERFL) complex at Texas Technology University. The building is a low-rise rotatable building with a height of 4 m, a width of 9.1 m and a length of 13.7 m. The fetch is considered long grass with a roughness height of 0.024 m [3]. Visualization measurements of the separation bubble and corner vortices and pressure on the roof were performed [4, 5, 6]. This yielded quantitative flow structure observations such as height and location of the reattaching separation bubble on the roof of the building.

From the smoke injection investigation, carried out by Sarkar *et al.* [5], it was observed that the separation bubble grew before collapsing and then reforming and this continued in a cyclical pattern. Observations from tuft visualization carried out by Wagaman *et al.* [6] indicated that there was little correlation of the size of the separation bubble to the wind speed or turbulence intensity. The separation bubble with the wind perpendicular to the long side of the building was 1.04 m [ranging from 0.73 - 1.22 m] high and 4.42 m [ranging from 3.20 - 5.06 m] long. The separation bubble height $H_c$, and length $X_c$ are defined as shown below in figure 2. Both Sarkar *et al.* and Wagaman *et al.* saw similar results for the flow perpendicular to the shorter side of the building, indicating that the flow can be considered two-dimensional when the flow is perpendicular to the face of the building.

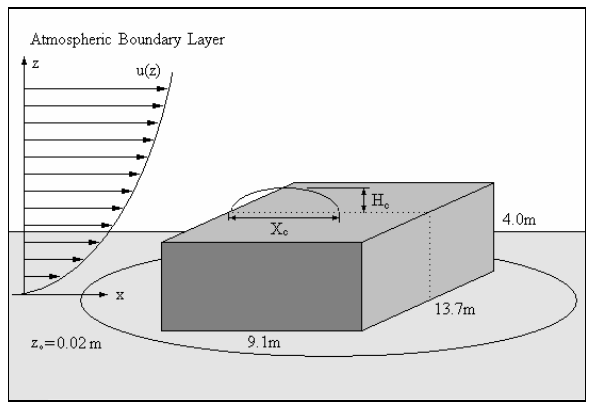


Figure 2: Building configuration with separation bubble defined.

Efforts have been made to model the WERFL building and reasonable results have been obtained with moderate grid sizes using k-$\varepsilon$ turbulence models [3, 7, 8, 9]. In these studies, pressure profiles over the length of the building were used as an indicator of the accuracy of the numerical model. Reasonable pressure profiles around the building were obtained from these studies when compared to the measured data.

Cowan *et al.* [10] examined the numerical accuracy of the k-$\varepsilon$ model with flow around buildings. In their assessment, independent groups modelled flow over buildings and dispersion models using commercial code and standard numerical modelling practices. It was determined that the numerical results from independent developers varied dramatically between one another and were highly dependent on mesh design and the advection scheme used. Coincidently, the results for a course mesh were sometimes more accurate than using a refined grid.

The poor performance of the k-$\varepsilon$ model is perhaps unsurprising given the high streamwise strain rates and highly anisotropic conditions [11]. Reynolds Stress Models (RSM) have shown better performance for free shear flows with strong anisotropy and flows with a sudden change in the mean shear rate making the model a good candidate for this flow configuration. RSM models have difficulty converging and are more computationally expensive [12].

## 3. NUMERICAL SIMULATION

Due to the uncertain reliability of the k-$\varepsilon$ models ability to successfully simulate flow over buildings, the current objective is to assess how applicable the k-$\varepsilon$ turbulence model is in resolving the flow over a building. Due to the availability of a relatively large amount of experimental data obtained at the WERFL building, it will be used as a base case to verify the results.

As previously mentioned, the flow around the WERFL building, with the wind normal to the long face, can be approximated as two-dimensional. In order to simplify the simulation, a two-dimensional model of the WERFL building is used as a base case to verify the numerical results. An unstructured grid is used with refined elements around the roof of the building and the ground. These elements expand away from the specified nodal lengths at a rate of 1.05. Recommendations specified by Hargreaves and Wright [13] concerning boundary conditions were applied. Inlet boundary conditions for an atmospheric boundary layer were given by [13] as:

$$u = \frac{u_*}{\kappa} \ln\left(\frac{z + z_0}{z_0}\right)$$

$$k = \frac{u_*^2}{\sqrt{C_\mu}}$$

$$\varepsilon = \frac{u_*^3}{\kappa\left(z + z_0\right)}$$

Where $u_*$ is the friction velocity, $\kappa$ is the von Karman constant (0.41), z is the height above the surface and $z_0$ is the aerodynamic surface roughness, in this case 0.02 m. Symmetry conditions were imposed at the top of the domain. A first order upwind differencing scheme was used. The commercial code ANSYS/CFX 11.0 was used for this study.

In order to accurately model the ground roughness a relationship between roughness specified in commercial software, based on sand grain particle roughness, and aerodynamic surface roughness is required. This has often been neglected in many previous investigations but is necessary to properly model the surface roughness. This relationship has been investigated by Hargreaves and Wright [13] and they concluded that the relationship $k_s$=29.6$z_0$ is valid for the commercial software used in this study, where $k_s$ is the roughness height based on sand particles and $z_0$ is the aerodynamic roughness.

From supportive literature, Cowan et. al. [10] recommends that the upwind and downwind dimensions should be 5H and 15H +-4H respectively, where H is the building height. In this analysis, the height H was defined as a characteristic height $H_c=B_s^{0.66}B_L^{0.33}$ where $B_s$ and $B_L$ are the respective smaller and larger dimensions of the building windward face. In the subsequent simulations a domain sized with a fetch of 7H, a height of 8H and a downwind dimension of 20H was applied. The domain dimensions are shown below in figure 3.

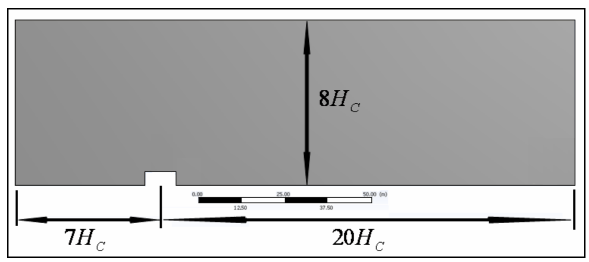


Figure 3: Domain used in the subsequent analysis.

To simplify the atmospheric conditions, it is assumed that the atmosphere is stable, i.e. the buoyancy turbulence generation is negligible compared to the mechanical turbulence production. Also, it is assumed that the buildings are smooth and the flow is fully turbulent.

## 4. RESULTS

To justify the applicability of the k-$\varepsilon$, the results must be physically realistic and thus must agree with full scale experimental results. To obtain accurate results the numerical solution must be independent of the choice of domain and grid size given that these constraints are satisfactory prescribed. Typical grid sizes for the building configuration were initially chosen from previous studies. Subsequent grid refinement was done to determine grid independence. Each consecutive grid refinement was produced by halving the number of specified cells over the building and ground while retaining the specified expansion ratio in the domain.

An initial grid was generated with the cell length of $0.01H_c$ along the top of the building and $0.1H_c$ along the ground. These successive grid refinements led to significant changes in the flow structure around the building. A summary of the results is shown in figure 4, indicating the reattachment location on the roof of the building for the k-$\varepsilon$ model. With the initial coarse grid, the separation bubble formed and was quite accurate in size and shape for the upwind scheme. A further refinement of the mesh led to a complete change in the flow structure with no reattachment zone on the roof of the building. Subsequent grid refinement led to a reattachment on the roof but over predicted the roof separation bubble reattachment location. Further grid refinement leads to quite accurate results with the solution attaining a reattachment location within the experimental variation. The resulting velocity field around the building for the final grid refinement is presented in figure 5.

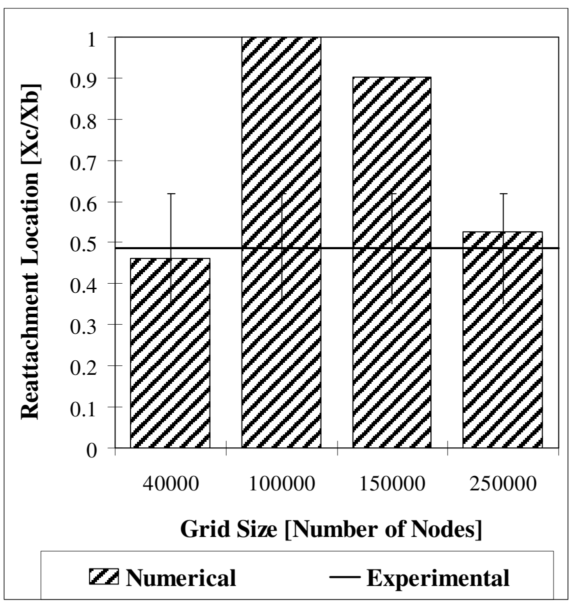


Figure 4: Non-dimensional reattachment locations on the roof of the building where Xc is the reattachment location and Xb is the building depth. Error bars indicate the experimental variation.

The pressure profiles obtained for the final grid refinement were quite similar to the experimental measurements and numerical results obtained in other investigations. The pressure distribution is plotted down the centre of the building and is displayed in figure 6. The non-dimensionalized pressure coefficient is the pressure nominal to the building divided by the dynamic pressure equivalent to, $\frac{1}{2}\rho V_H^2$ where ρ is the density of air and $V_H$ is the free stream velocity at the height of the building. The distance along the x-axis begins at the bottom centre of the windward wall and progresses over the building to the bottom centre of the leeward side. Experimental results were obtained from Paterson and Holms [8]

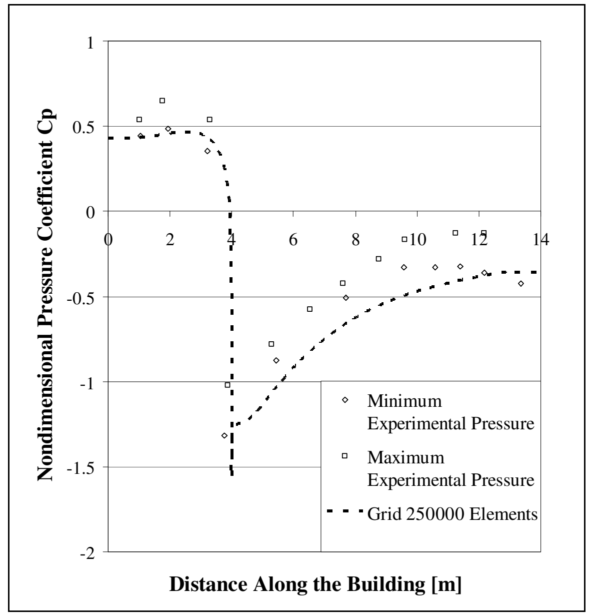


Figure 6: Pressure distribution over the building for the finest grid compared to experimental data.

Transient solutions were run on the highest refined grid and reattachment settled to steady oscillations after 10 seconds. Oscillations were small with a peak-to-peak range of 0.32 m and a period of two seconds with a mean value equivalent to the steady state solution. The transient response is expected as the flow is unsteady indicated by the range of values for reattachment locations and pressure distribution

The domain size was doubled in height and doubled in downstream length to test for domain dependence. It was seen that the size of the domain had little effect on the results over the building and thus can be regarded as independent of the solution.

## 5. CONCLUSIONS

From the results presented above, it is seen that the numerical solution obtained from the k-$\varepsilon$ turbulence model can be misleading. As the grid is refined, the solution diverges from the experimental results and then converges on the correct result. The solution is grid dependent since the solution does not converge on a solution as the grid is refined. Any subsequent refinement may lead to a converging solution but any application of the grid independent solution would be too numerically expensive in even the simplest two-

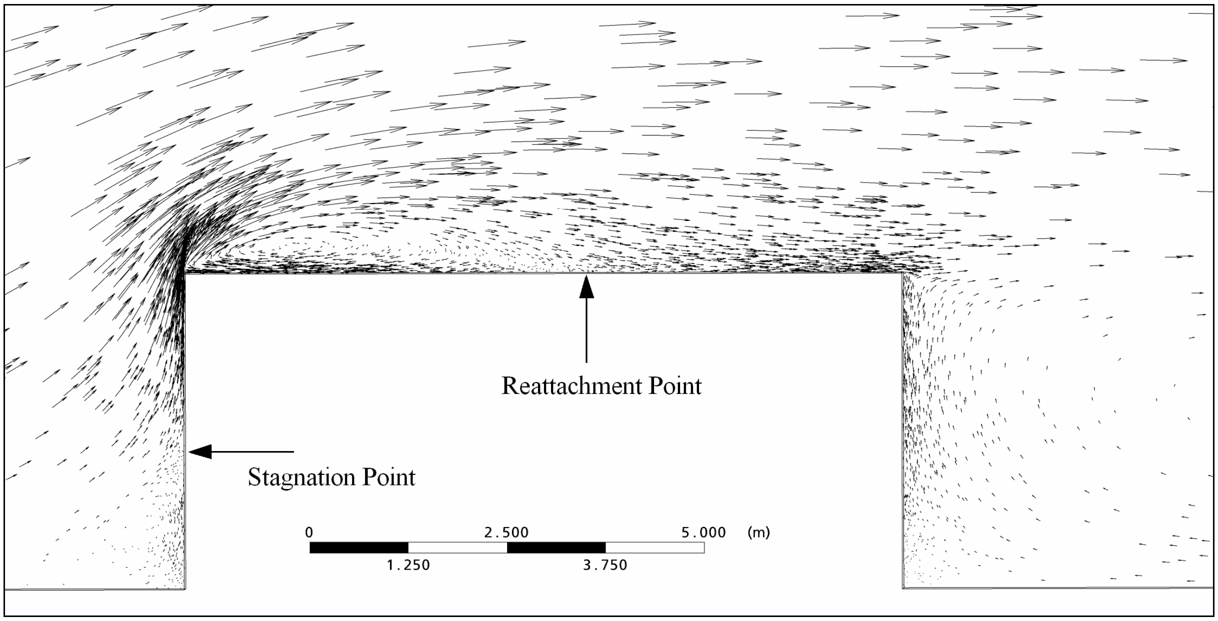


Figure 5: Velocity profile around the building for the finest grid with the numerical prediction of the stagnation point and reattachment location identified.

dimensional cases. At present, applying the k-$\varepsilon$ model to any complicated geometry such as three-dimensional flow through the urban environment would be far too computationally expensive and thus impractical.

The presented work emphasises the strong dependence of the numerical solution on grid size. If the grid is not verified over a large range of nodal refinements, conclusions may be drawn for grid dependent solutions. This is difficult to verify especially for three-dimensional problems since the studied range of grid refinements would be very computationally expensive. It is seen that convincing results can be obtained even with a quite coarse mesh as in the first grid studied. These results suggest that proposed converged solutions from previous investigations may be grid dependent.

Recent investigations carried out by Hargreaves and Wright [13] as well as Blocken et al. [14] have indicated many of the difficulties with sustaining a atmospheric boundary layer profile over a uniform terrain in commercial software. Attempts were made to minimize these effects in this study by making the fetch as short as possible in front of the building in the domain. These effects of the sustainable boundary layer are currently being investigated on the previously investigated building configurations.

## 6. Recommendations and Outline for Future Work

Two-equation models are an attractive turbulence model due to their simplicity and relative robustness. Inconsistent results are obtained when the k-$\varepsilon$ turbulence model was applied to this simplified case of flow over a two-dimensional building. The upwind advection scheme produced an accurate result but was misleading in the fact that the results were not verified to be grid independent. Even though the k-$\varepsilon$ model is capable of solving the flow accurately for grid dependent solutions, any implementation in complex geometries in three-dimensions would be impractical due the increase in computational expense when implanted in three dimensions.

Aside from the k-$\varepsilon$ turbulence model, alternative turbulence models should be considered for their usefulness in this problem. It is suggested that models that perform more favourably with high stream-wise strain rates and in anisotropic flow be investigated such as the Reynolds shear stress models. Currently, the k-$\omega$ and the blended k-$\varepsilon$ and k-$\omega$ models are being studied with the Reynolds shear stress model.

## ACKNOWLEDGEMENTS

The authors would like to recognize the funding from the Ontario Centre of Excellence (OCE).

## REFERENCES

[1] K.C. Kim, H.S. Ji, S.H. Seong, Flow structure around a 3-D rectangular prism in a turbulent boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics,* 91:653-669, 2003

[2] S. Mertens, The energy yield of roof mounted wind turbines. *Wind Engineering*, 27:507-518, 2003

[3] R.P. Selvam, Computation of pressures on Texas Tech Building. *Journal of Wind Engineering and Industrial Aerodynamics*, 41-44:1619-1627, 1992

[4] D. Banks, R.N. Meroney, P.P. Sarkar, Z. Zhao, F. Wu, Flow visualization of conical vortices on flat roofs with simultaneous surface pressure measurement. *Journal of Wind Engineering and Industrial Aerodynamics*, 84:65-85, 2000

[5] P.P. Sarkar, Z. Zhao, K. C. Mehta, Flow visualization and measurement on the roof of the Texas Tech building. *Journal of Wind Engineering and Industrial Aerodynamics*, 69-71:597-606, 1997

[6] S.A. Wagaman, K.A. Rainwater, K.C. Mehta, R.H. Ramsey, Full-scale flow visualization over a low-rise building. *Journal of Wind Engineering and Industrial Aerodynamics*, 90:1-8, 2002

[7] H. Jianming, A.S. Song, A numerical study of wind flow around the TTU building and the roof corner vortex. *Journal of Wind Engineering and Industrial Aerodynamics*, 67&68: 547-558, 1997

[8] D.A. Paterson, J.D. Holms, Computation of wind pressures on low rise structures. *Journal of Wind Engineering and Industrial Aerodynamics,* 41-44:1629-1640, 1992

[9] T. Stathopoulos, Computational wind engineering: Past achievements and future challenges. *Journal of Wind Engineering and Industrial Aerodynamics,* 67&68:509-532, 1997

[10] I. R. Cowan, I. P. Castro, A.G. Robins, Numerical considerations for simulations of flow and dispersion around buildings. *Journal of Wind Engineering and Industrial Aerodynamics,* 67&68:535-545, 1997

[11] B.E. Lauder, Second-moment closure and its use in modeling turbulent industrial flows. *International Journal of Numerical Methods Fluids,* 9:963-985, 1989

[12] ANSYS CFX, CFX 5.7.1 Solver Theory, ANSYS Inc., Ontario Canada, 104-105, 2004

[13] D.M Hargreaves., N.G. Wright, On the use of the k–$\varepsilon$ model in commercial CFD software to model the neutral atmospheric boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics,* 95: 355-369, 2007

[14] B. Blocken, T. Stathopoulos, J. Carmeliet, CFD simulation of the atmospheric boundary layer: wall function problems. *Atmospheric Environment*, 41:238-252, 2007