A Numerical Analysis of the Updraft Over a Building Model

Abdulghani Mohamed* - *s3165510@student.rmit.edu.au*

Simon Watkins* - *simon@rmit.edu.au*

* *School of Aerospace, Mechanical, and Manufacturing Engineering*

INTRODUCTION

Harvesting energy from the surrounding environment offers the potential to significantly increase range and endurance of Unmanned Air Vehicles (UAVs). The possibility of using naturally occurring thermals or updrafts as an energy source to gain height remains relatively unexplored. An early encouraging study by Allen, (2005) concluded that the endurance of a representative UAV could be increased by up to 12 hours by using thermal lift. Cutler et al., (2010) presented an important study into the feasibility of energy harvesting using orographic lift during intelligence, surveillance, and reconnaissance, (ISR) missions. This study showed that when an energy source (such as a slope) is within 400 m of the target, no propulsive power was required for the selected UAV to orbit a target for ISR (the platform could maintain height using the vertical component of the flow up the slope). The complex flow patterns that occur in suburban environments are typical for UAV and Micro Aerial Vehicle (MAV) operations. These flow patterns can aid the vehicles to gain height and soar further or even recharge on-board batteries through regeneration. Understanding the flow patterns around buildings in suburban environments with particular attention to the updraft of airflow upstream of the building's roof top is therefore essential for UAV/MAV operations and energy harvesting applications.

Numeric models of the turbulent Atmospheric Boundary Layer (ABL) have been implemented for studying and analysing building envelopes, natural ventilation, wind loading, dispersion of air pollutants and other flow predictions (Tutar and Oguz, 2002). However, few if any studies have focused primarily on updrafts over rooftops. Most numerical studies focused on the general flow around single building models (Baskaran and Stathopoulos, 1989; Stathopoulos and Zhou, 1993; Paterson and Apelt, 1990) where the standard k-ε viscous turbulence model was implemented. Murakami et al, (1990, 1993); He and Song, (1992) used the Large Eddy Simulation approach. From these studies, the LES model seems to accurately predict the flow behavior compared to the other models. Hence, to develop an understanding of the energy potentially available near the tops of buildings for endurance extension through soaring, velocity magnitudes have been mapped in the region where updrafts are expected. Characterization of this updraft field provides an indication of the energy availability for harvesting and inform UAV/MAV configuration and design.

METHODOLOGY

The representative building selected for the study presented in this paper is Building 201, (43 meters high and 38 meters wide) of RMIT University's Bundoora Campus (Melbourne Australia). The buildings' unique position and environment matched the topography of a suburban terrain. In a separate paper by White et al., (2011) a 1/100th scale model of Building 201 was used for wind-tunnel testing and the results were validated by measurement from the roof of the actual building. The data from the wind-tunnel study are used for validation of the CFD results. The numerical study was conducted in 2D as a steady state problem, which then evolved into a transient 3D study. The 2D study allowed careful inspection of grid performance and domain size, which provides a basis for the 3D study. The 2D study has been simulated using the standard k-ε model with enhanced wall treatment, while the 3D study used the Large Eddy Simulation approach using the Smagorinsky-Lilly model for sub-grid scale. 2D analysis has some inherent limitations where the 3D effects are neglected assuming the cross section being analysed is infinitely wide. This results in some inaccuracy of the results since the flow around the building is expected to be highly three-dimensional. The setup and construction of the computational model is outlined by Mohamed et al., (2012a, 2012b).

RESULTS AND DISCUSSIONS

Two-dimensional Results

The flow features of the simulation show agreement with predicted behavior and work previously published by researchers. Please note that all the presented results are normalised to the buildings reference height, H_b. Consequently the scales of the contours can be viewed as velocity ratios to the wind speed at H_b. As predicted, the updraft region contains the highest magnitude of velocity. The y-axis velocity contour shows the y component of the flow's velocity near the rooftop. With the zero velocity clearly identified on the contour, it can be seen where there are updrafts and down-drafts. Regions with strong updrafts are clearly visible in Figure 1, which represent a region of interest for UAV/MAV flight.

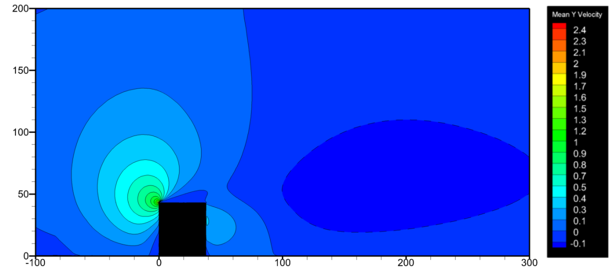

Figure 1. 2D averaged y-velocity contours in ms^{-1}

Three-dimensional Results

The flow features of the 3D simulation also show agreement with the work previously published by researchers, as the basic flow features were replicated. As predicted, the updraft region contains the highest magnitude of vertical velocity at the roofs edge. Figure 2 shows the velocity contours. It's important to note that the contours are positioned on the lateral center of the building (i.e. at z = 0). In order to visualize the 3-Dimensionality of the updraft region and its core strength, an iso-surface was created showing 4 different core intensities (see figure 3).

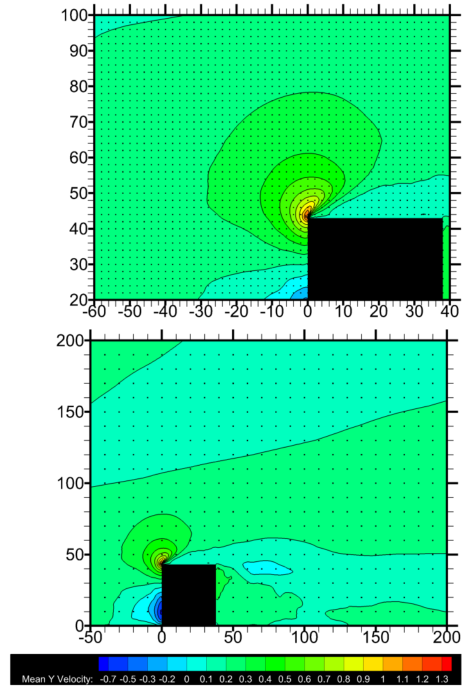

Figure 2. 3D mean vertical velocity contours.

Wind-tunnel Comparison

The same building geometry was tested in the wind tunnel at 1/100th scale with similar velocity and turbulence intensity profiles as presented by White et al., (2011). The wind-tunnel experiment used cobra probes to measure the velocity vectors in a spacial matrix in the vicinity of the building's rooftop. The same matrix was created in the domain of the numeric study for vector magnitude and direction comparison as illustrated in figure 4. Both sets of results presented have been normalised to H_b.

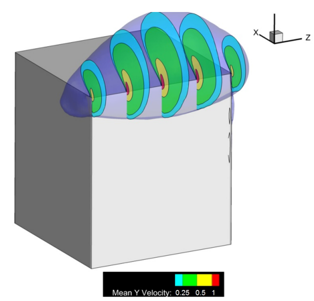

Figure 3. Iso-Surface showing 3 levels of mean vertical velocity.

It was observed that for the majority of the results, the difference was below 20%. This difference was expected because of a number of reasons. The velocity profile tested in the wind tunnel had a slightly varied shape compared with the theoretical profile used by the numerical analysis. The variation was also partially due to the roughness elements installed in the wind-tunnel to replicate the ABL. The wake from those roughness elements also affected the stagnation location on the face of the building as observed from figure 4, where the vectors at a height of 34.5m show almost stagnant flow in the case of the experimental results. Even the simulated Reynolds Number tested in the wind-tunnel was different, further contributing to the variation of results. The magnitude of the vectors is also different because the free stream velocity was about 3 times higher in the wind-tunnel experiment, which will also affect the flow angle upstream. Difficulties in numerical simulation of turbulent flow around buildings is another contributing factor.

CONCLUSIONS

The CFD model has accurately represented the flow behaviors as previously published. The 3D analysis showed significant vortex shedding and highly turbulent flow entering the domain which was a phenomenon that wasn't captured by the 2D case. When comparing results it is evident that the 2D case over-predicted the updraft region while the 3D case provided results more representative of those obtained from wind-tunnel testing. The LES approach gave reliable results compared with the *k-ε* model. It is hence a recommendation for further progress, that a finer mesh resolution should be used to improve the LES results in addition to testing various building configurations.

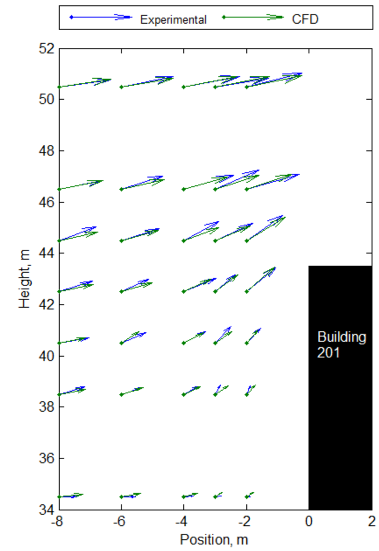

Figure 4. Velocity vector comparison.

REFERENCES

Allen, M. J. (2005), *Autonomous soaring for improved endurance of a small uninhabited air vehicle*, AIAA No. 2005-1025

Baskaran A., Stathopoulos T. (1989), *Computational evaluation of wind effects on buildings*, Building Environment 24: 325-333.

Cutler M. J., McLain T. W., Beard R. W., Capozzi B. (2010), *Energy harvesting and mission effectiveness for small unmanned aircraft*, AIAA

He J., Song C. C. S. (1992), *Computation of turbulent shear flow over surface mounted obstacle*, ASCE Journal of Engineering Mechanics

Mohamed, A., White, C., & Watkins, S. (2012a), *A numerical study of the updrafts over a building, with comparison to wind-tunnel results*, 15th Australasian Wind Engineering Society Workshop. Sydney, Australia, 23-24 February, 2012

Mohamed, A., White, C., Watkins, S. and Carrese, R. (2012b), *Energy harvesting from updraft over buildings by micro aerial vehicles*, Journal of Wind Engineering and Industrial Aerodynamics. (Submitted for Publication)

Murakami S. (1993), *Comparison of various turbulence models applied to bluff body*, Journal of Wind Engineering and Industrial Aerodynamics

Murakami S., Mochida A. (1990), *A 3-D numerical simulation of air flow around a cubic model by means of the k-ε model*, Journal of Wind Engineering and Industrial Aerodynamics

Paterson D. A., Apelt C. J. (1990), *Simulation of flow past a cube in a turbulent boundary layer*, Journal of Wind Engineering and Industrial Aerodynamics

Stathopoulos T., Zhou Y. S. (1993), *Numerical simulation of wind-induced pressures on buildings of various geometries*, Journal of Wind Engineering and Industrial Aerodynamics

Tutar M., Oguz G. (2002), *Large eddy simulation of wind flow around parallel buildings with varying configurations*, Fluid Dynamics Research.

White C., Lim E. W., Watkins S., Mohamed A., Thompson M. (2011), *A feasibility study of micro air vehicles soaring tall buildings*, Journal of Wind Engineering and Industrial Aerodynamics 103: 41-49.