

A CFD investigation into the wind flow around several generic building types situated within a built environment.

Thomas Rogers, Saffa Riffat and Siddig Omer

Institute of Sustainable Energy Technology, University of Nottingham, UK,
laxter@nottingham.ac.uk

ABSTRACT: This paper presents results of CFD modelling carried out to investigate the wind flow around the rooftops of several kinds of homogenous urban building structures. The intention being to explore the optimal positioning of micro-scale building-mounted wind turbines. Buildings examined include large flat-roofed commercial properties (industrial estates, business parks, commercial tower blocks) and pitched-roofed residential properties (terraced, semi-detached and detached houses, residential tower blocks). A method for simplifying and modelling the flow around buildings was developed and is presented here. Typical turbulence intensity levels are also investigated. Several conclusions were drawn from the results; these suggest that, depending upon local site conditions, commercial buildings are better suited to building mounted turbines, it is inadvisable to mount a turbine below the ridgeline of pitched roof dwellings and speed-up effects are greatly reduced for buildings situated within the urban environment.

Keywords: wind energy, micro-generation, CFD, urban environment.

1. INTRODUCTION

Micro-generation is generally acknowledged as one of the sustainable technologies able to help decrease humankind's dependency on hydrocarbon energy sources. Which particular form of micro-generation to choose depends upon the local energy sources available. The more choices available, the more likely it is that energy demands can be supplied. Wind power is perceived as being one of these options. However, in comparison to other micro-generation technologies (solar thermal and photovoltaic, heat pump technology, biomass boilers) it is still in its fledgling period. Previous research into urban wind power has highlighted several areas lacking knowledge [1] and one of the

main areas was the absence of data on the urban wind resource around rooftops. It is the aim of this paper to investigate and contribute to this knowledge base using CFD.

2. METHODOLOGY

Computational fluid dynamics (CFD) has been used in a variety of applications over recent years, including the study of wind flow through urban street canyons, pedestrian comfort levels, the study of ventilation effects, dispersion modelling, and the study of the effects of wind flows on tall buildings. Although the potential for CFD is undoubtedly great and its uptake by industry and academics is on the increase, it is acknowledged that caution must be

applied when applying it. Castro and J.M.R. Graham [2] report that problems arise if CFD is used:

- without a sound understanding of the fluid mechanics appropriate to the particular problem being investigated,
- an awareness of the extent to which the code being used has been validated for similar problems
- and a clear understanding of the sources of uncertainty and the accuracy levels actually needed

For these reasons, numerous sources have been used when constructing the geometries, setting up the numeric modelling and interpreting the CFD calculations for this investigation. Of greatest assistance to this process was the domain creation and numeric modelling techniques outlined in the COST best practice guidelines published in May 2007 [3]. Other useful guidelines were found in Sander Mertens, Wind Energy in the Built Environment [4], in which selection of appropriate turbulence models are discussed.

2.1 Turbulence

Although the calculation of wind speeds are paramount when assessing the best site for a wind turbine, turbulence is also a significant contributing factor. Previous work has found that turbulence considerably reduces the performance of a horizontal axis wind turbine (HAWT) [5]. Turbulence intensity, sometimes referred to as the turbulence level, is defined as:

$$I \equiv \frac{u'}{U}$$

Where u' is the root-mean-square of the turbulent velocity fluctuations and U is the mean velocity. Turbulence intensity levels of 50% can reduce the potential power yield by around 14%. As well as this, fatigue loads created by the inertia of the rotor blades and the load forces exerted on the turbine tail fin by a fluctuating wind direction, will substantially reduce the turbines lifespan. Vertical-axis wind

turbines (VAWTs) of the savonius type are less affected by turbulence. However their aerodynamic efficiency, or ability to extract power from the wind, is substantially less than for a HAWT. The ANSYS Fluent CFD software [6] is able to provide turbulence intensity values and these are presented and discussed in section 3 of this paper.

2.2 Building selection

Tests were carried out on a number of different building configurations, chosen by considering:

- urban environments with homogenous building types,
- building types targetted by wind turbine manufacturers and installers (the domestic market),
- and, building types with the greatest potential for the installation of wind turbines (tall buildings).

Unlike other studies currently in existence which concentrate on designing buildings suitable for wind turbine integration, this paper will concentrate on existing building stock and consider the wind turbines in their retrofit state.

The geometries of the selected buildings were hugely simplified when compared with the real urban environment. Only the basic buildings themselves were modelled; vegetation (trees, hedges, etc), fences, garages, walls, chimneys, etc were ignored as including these features in the geometry would increase the computational demand beyond it's current ability.

The COST guidelines [3] suggest that any obstacle up to a distance of six times the height of the building under observation would effect the approaching flow. This suggestion was used to determine the number of buildings to include in the flow problem. For all simulations, the target building was placed in the centre of a row at the downwind position of an array of 7 by 7 buildings.

Freestream wind speeds of 10m/s are used in each simulation. Building dimensions and spacings are shown in

Figure 1. These dimensions were found by sampling and averaging dimensions from real urban case studies. In all cases of CFD geometries, the buildings have 0 surface roughness whilst the ground has a surface roughness length of 0.1. Simulations are performed for 4 different approach angles; at 0°, 30°, 60° and 90°.

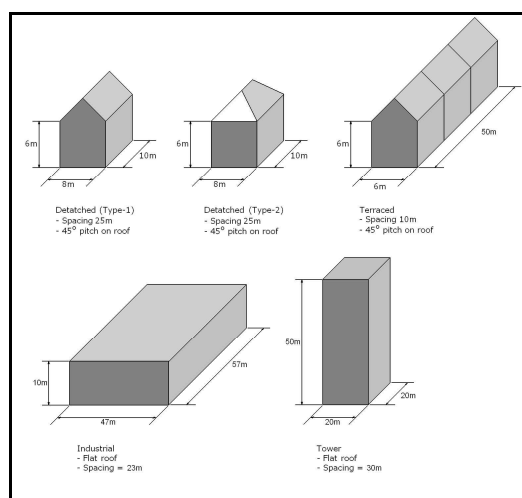


Figure 1. Building dimensions and spacing.

2.3 Domain creation

The building array was placed a distance of five of times the building height (5H) from the inlet face and 10H from the outlet flow. The sides and domain ceiling were a distance of 5H from the array. The grid resolution was weighted to take into account that the region of interest is within 5m of the roof surface. At the roof surface the cell size is set at 0.2 and this increased by a factor of 1.2 moving away from the roof surface. To keep the number of cells in the domain to a minimum, and consequently CPU time, only the target building and buildings directly upwind were given sizing functions. This limited the domain size to 2 million cells for the domestic properties and 3 million cells for the larger commercial properties.

2.4 Numerical modelling

For this body of work, the Reynolds Averaged Navier Stokes (RANS) equations were used along with the κ - ϵ realized turbulence model. This turbulence model is

a feature of Fluent 6.1 [6] and is a more elaborate version of the standard κ - ϵ model. It has additional terms and functions in the transport equation for κ (turbulent kinetic energy) and ϵ (turbulent dissipation rate). These terms give better results for a wider range of flows including separation and recirculation, both types of which are prevalent for the flows examined here. Current literature suggests this to be the most favoured turbulence model [4]. Alterations to the CFD solver were made as suggested by Richards and Hoxey [7]. These alterations, to several model constants, improved the model accuracy when Richards and Hoxey compared their κ - ϵ turbulence model with detailed readings of the pressure and flow field around a 6m cube positioned in open air.

2.5 Flow initiation

The approaching wind profile is an important part of any wind flow problem. Failure to include one can invalidate the results. It is generally accepted that the log law provides the most accurate wind profile (wind speed versus height) assuming a suitable surface roughness value is provided. Previous work has explored the creation of a user defined function (UDF) to create a log law profile for the approaching wind profile, entering the domain at the inlet face. However this flow structure deteriorates within the first few meters of the inlet face. For this body of work, the approaching wind profile was created within the domain itself by the use of evenly spaced rectangular obstacles, half the height of the building under observation.

3. RESULTS AND DISCUSSION

In order to process and interpret the substantial numerical results, a technique for representing the data in an easily interpreted format is presented. Plan views of the target buildings are used to show the best siting. The plan view shows the averaged wind

speed at a height of 2m above the roof of the building. This is the realistic hub heights for both HAWTs and VAWTs. Results for wind flow and turbulence investigations for each type of building are presented in Figure 3.

3.1 Detached property (type 1)

Results at 0 degree incident flow angle show either end of the ridgeline to be the better place to site a turbine. This is applicable to this particular array of buildings as the lower wind speeds experienced towards the centre of the roof for a wind direction of 0° are due to shadowing effects of the building immediately opposite and upwind.

The wind speed tends to increase as the wind direction becomes parallel with the roofline. In order to take advantage of the

higher wind speeds found at angles of 90° and 270° these results suggest to site the turbine at the centre of the ridgeline.

Perhaps predictably, it would be unwise to site a turbine on the lee side of prevalent wind direction. Wind speeds here tend to be on average 2 m/s lower than the windward side. Figure 2 suggests that the overall wind speeds are considerably reduced by this configuration of buildings. The maximum wind speed experienced (5m/s) being just half of the free stream wind speed. Speed-up effects popularly quoted as existing in the built environment are not found here. Results of turbulence modelling (Figure 3) show the turbulence intensity levels around the rooftops to be well above those levels recommended for horizontal-axis turbines.

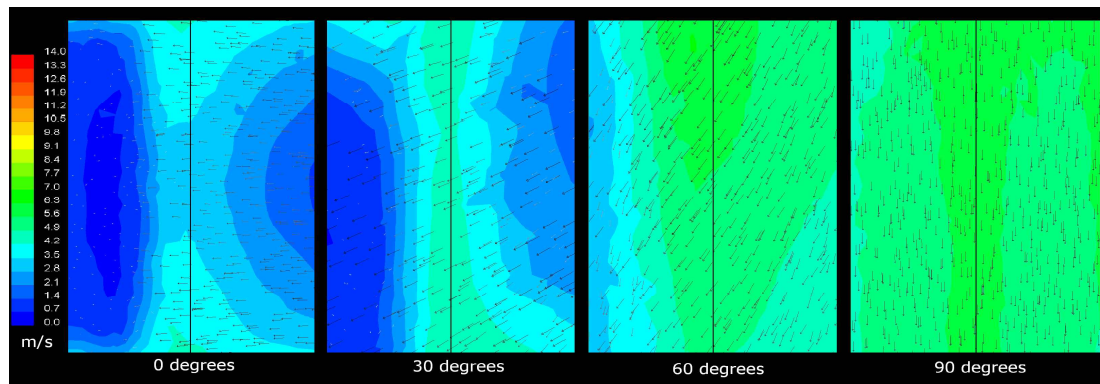


Figure 2. CFD results for the wind flow over type-1 pitched-roof detached property at varying approach angles (see Figure 1 for dimensions).

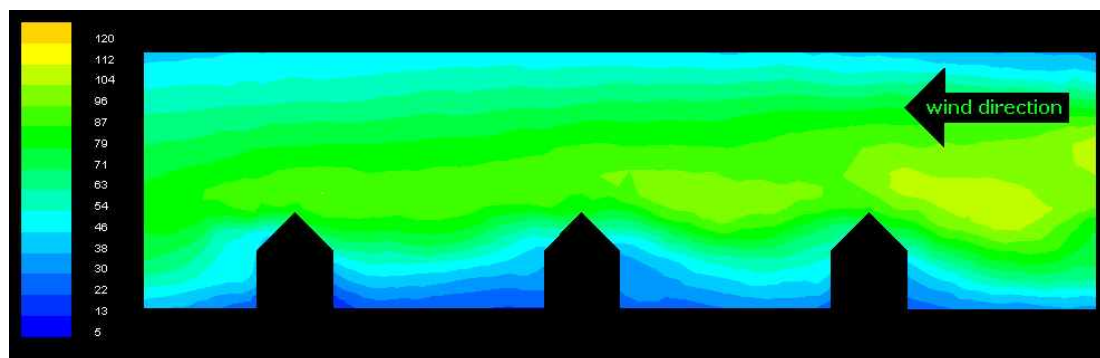


Figure 3. Turbulence intensity profile (%) of the wind flow around type-1 pitched-roof detached properties (see Figure 1 for dimensions).

3.2 Detached property (type 2)

Results indicate that a suitable position to site a turbine would be in the

centre of the roof on the ridgeline. Wind speeds remain similar independent of the wind direction. However wind speeds are

still approximately half of the freestream wind speed.
When compared with the type-1 detached property, the type-2 property experiences less turbulence when the incident flow

angle is perpendicular to the roofline but more when the angle is parallel to the roofline, this is also when the wind speed is higher.

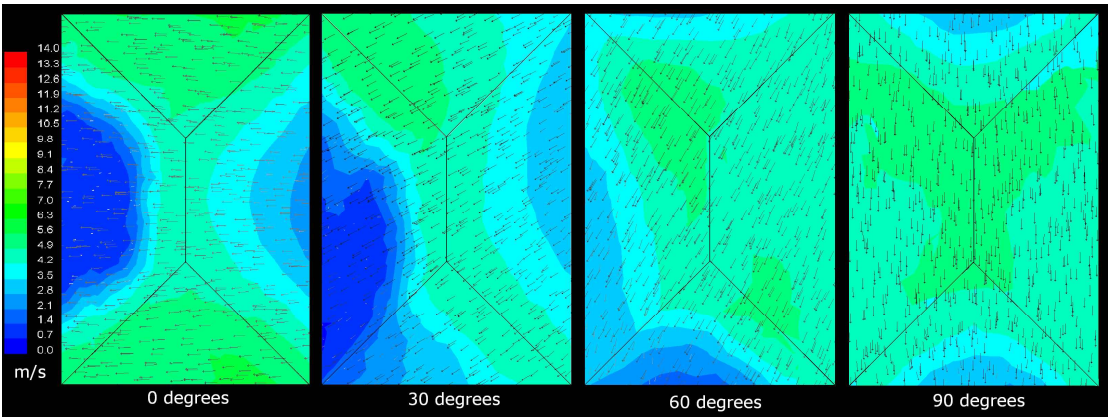


Figure 4. CFD results for the wind flow over type-2 pitched-roof detached property at varying approach angles (see Figure 1 for dimensions).

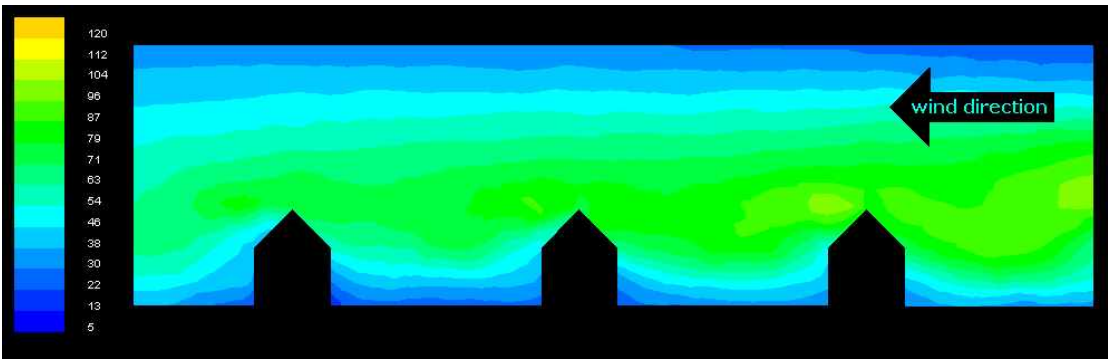


Figure 5. Turbulence intensity profile (%) of the wind flow around type-2 pitched-roof detached properties (see Figure 1 for dimensions).

3.3 Terraced housing

The wind speeds for terraced housing are substantially lower than for other domestic properties when the wind is perpendicular to the ridgeline. Figure 7 shows that this is also the case for the turbulence intensity. Flows perpendicular to terraced housing returns the highest

values for turbulence. However, when the flow is parallel to the ridgeline, higher wind speeds and lower turbulence exists. The turbulence is still higher than recommended for horizontal-axis turbines and wind speeds are still less than half the freestream speed.

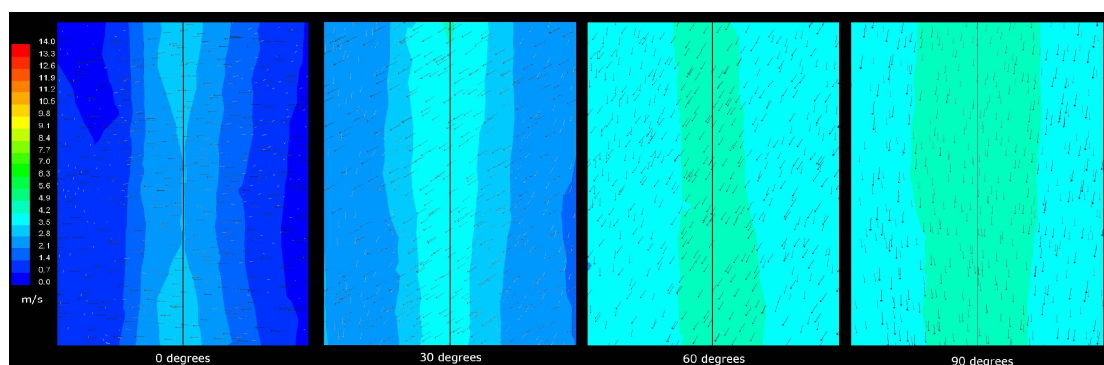


Figure 6. CFD results for the wind flow over a pitched-roof terraced property at varying approach angles (see Figure 1 for dimensions).

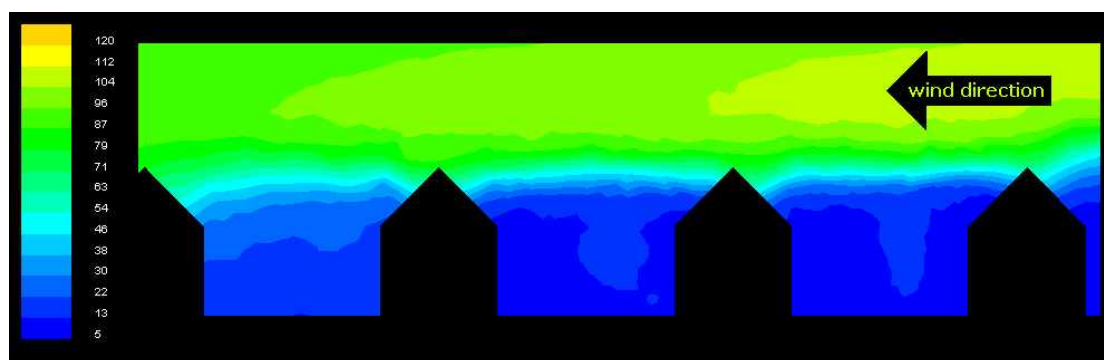


Figure 7 Turbulence intensity profile (%) of the wind flow around pitched-roof terraced properties (see Figure 1 for dimensions).

3.4 Industrial buildings

Figure 8 shows the wind speeds for the wind flow over flat-roofed industrial type buildings. The wind speeds are higher than for the domestic properties described previously (~75% of the freestream wind speed rather than 50%). The higher wind speeds found at the leading edge of the building also coincide with the higher turbulence levels (Figure 9). Flow reattachment on the lee-side of the roof produces unturbulent, laminar flow,

approximately half that of the freestream speed. This flow profile would be suitable for the siting of HAWTs. VAWTs, which are able to operate efficiently in omnidirectional wind flows found in turbulent conditions, would be able to take advantage of the higher wind speeds found at the leading edge. This knowledge, along with the prevailing wind direction, would assist in deducing the best site for a turbine on a large, flat-roofed, industrial building.

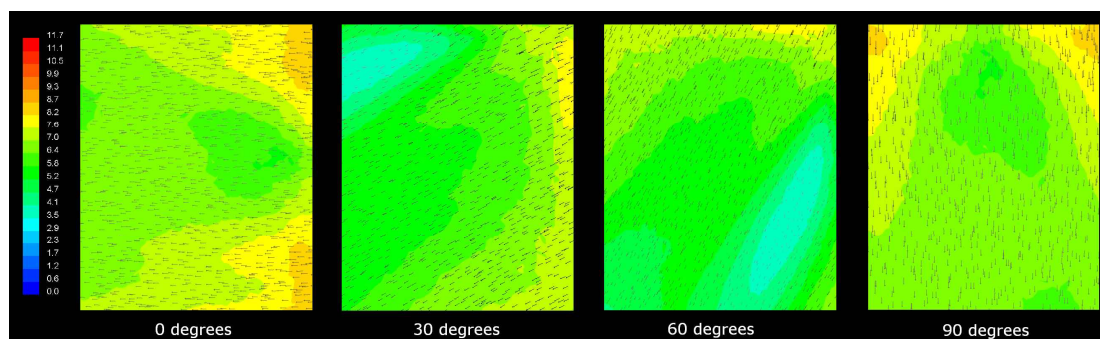


Figure 8. CFD results for the wind flow over a flat-roofed industrial building at varying approach angles (see Figure 1 for dimensions).

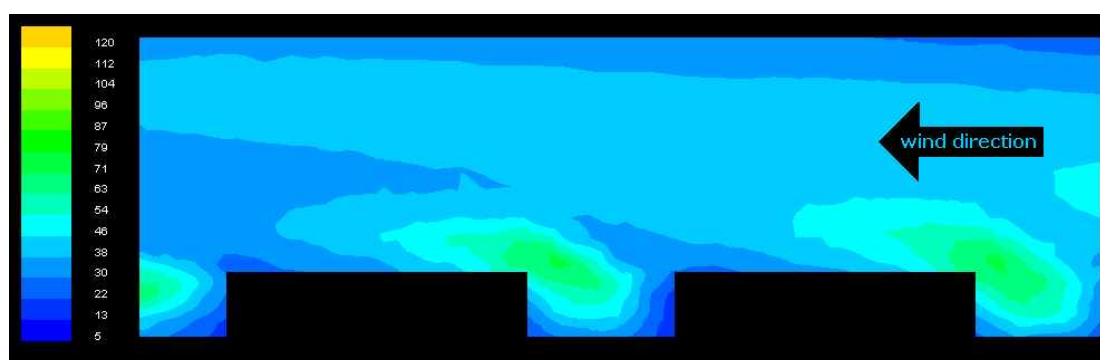


Figure 9. Turbulence intensity profile (%) of the wind flow around flat roof industrial buildings (see Figure 1 for dimensions).

3.5 Tower blocks

Tower blocks unsurprisingly return the best wind speeds, but only if they clear the height of surrounding buildings. High turbulence found at the leading edge of the roof (Figure 11) discourages the siting of a HAWT at this position. Judging from slides 1 and 2 in Figure 10 and taking into account the turbulence results summarised

by Figure 11, the optimum siting appears to be in the centre of the tower. Observing the wind profile from the site (Figure 10, slide 3), a speed-up effects exist at a height of 10m above the top of the building. This speed-up takes the wind speed above the free-stream position.

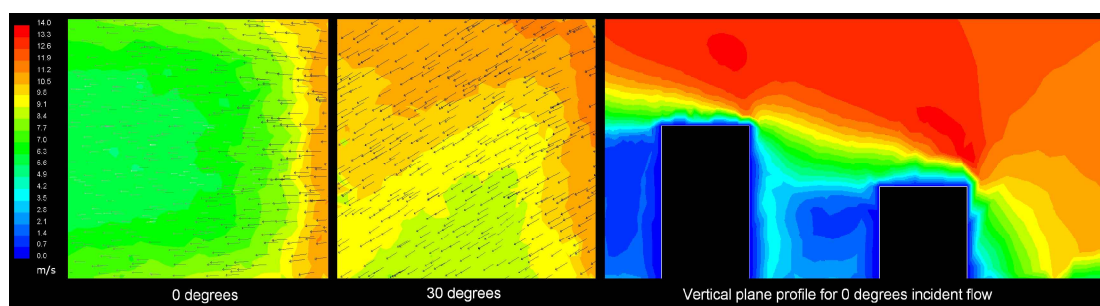


Figure 10. CFD results for the wind flow over a high-rise building at varying approach angles (slides 1 and 2) and the wind profile (slide 3).

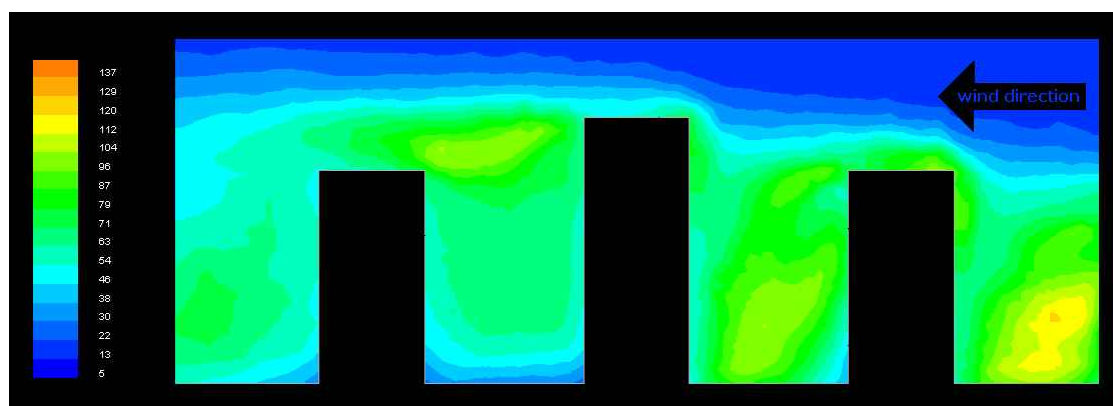


Figure 11. Turbulence intensity (%) for high-rise buildings (see Figure 1 for dimensions).

4. CONCLUSIONS

It could be concluded that wind speeds for domestic properties within urban areas are substantially less than the freestream velocity and suggest that, unless local site conditions are favourable, it would be inadvisable to mount micro-scale wind turbines in these areas. Exceptions include:

- open terrain around the house, especially in the prevailing wind direction,
- the prevailing wind direction is parallel to the ridgelines of multiple properties and vegetation is minimal.

Other options include mounting the wind turbine at the centre of the ridgeline.

Commercial properties with flat roofs or low angled pitched-roofs show reasonable wind speeds, although they are still approximately 75% of the freestream wind speed. If conditions are suitable then VAWTs could be used along the leading edge in the prevailing wind direction. High turbulence at the leading edge and flow reattachment towards the lee of the roof infer that HAWTs should only be considered if they are mounted towards the lee of the roof in the prevailing wind direction.

Acknowledgements: Project sponsored by EPSRC and E-ON UK.

5. REFERENCES

- [1] Dutton, A. Halliday, J, Blanch, M. 2005. The Feasibility of Building-Mounted/Integrated Wind Turbines: Achieving their potential for carbon emission reduction. Energy Research Unit. Carbon Trust Report. pp4-5. UK.
- [2] Castro, I, Graham, J. 1999. Numerical wind engineering: the way ahead? Structures & Buildings. Vol 134. pp275-277. UK.
- [3] Britter, R, Schatzmann, M. 2007. Model evaluation guidance and protocol document – Quality assurance and improvement of micro-scale meteorological models. COST Action 732. Germany.
- [4] Mertens, S. Wind Energy in the Built Environment. Multi-Science. pp53-64. 2006. UK.
- [5] Rogers, T. 2008. The effect of turbulence on microscale HAWTs. laxter@nottingham.ac.uk. UK.
- [6] Fluent 6.1, Fluent, Inc., ANSYS, Inc. www.fluent.com. 14th June 2008.
- [7] Richards, P, Hoxey, R. 1993. Appropriate boundary conditions for computational wind engineering models using the κ - ϵ turbulence model. Journal of Wind Engineering and Industrial Aerodynamics. Vol 46 & 47. pp145-153. UK.