Roof Mounted Wind Turbines:
A Methodology for Assessing Potential Roof Mounting Locations

ISLAM ABOHELA[1], NEVEEN HAMZA[1], STEVEN DUDEK[1],

[1]School of Architecture, Planning and Landscape, Newcastle University, Newcastle upon Tyne, UK

ABSTRACT: This paper is part of a research investigating variables affecting the performance of urban wind turbines, specifically roof mounted wind turbines. The aim of this paper is to present the results of the research focusing on the technology of wind turbines to be installed in the vicinity of buildings, the best practice guidelines for using CFD as a tool for assessing urban wind flow and the effect of some variables including wind direction, roof shape, building height and surrounding urban configuration on the energy yield and positioning of roof mounted wind turbines. Results show that for each roof shape there is an optimum mounting location for roof mounted wind turbines and among the investigated roof shapes, the barrel vaulted roof had the highest accelerating effect on wind flow above the roof. Also, it is evident that changing wind direction, building height and surrounding urban configuration had an effect on choosing the optimum mounting location and the energy yield.
Keywords: wind turbines, CFD, roofs, urban, energy

INTRODUCTION
The increasing interest among architects and planners in designing environmentally friendly buildings has led to a desire to explore and integrate renewable sources of energy within the built environment. Roof mounted wind turbines is a technology that presents a high potential for integration within the built environment. However, there is a state of uncertainty regarding the viability of these wind turbines. This paper argues that part of this uncertainty is attributed to:

- Choosing inappropriate wind turbines technology to operate in the vicinity of buildings.
- Lack of accuracy in assessing the in-situ wind conditions for the proposed mounting location.
- Uninformed decisions about positioning and mounting urban wind turbines.
- Lack of consideration to the wind accelerating effect of different roof shapes, buildings' heights and surrounding urban configurations.

This paper aims to present the results of a research by the authors tackling the integration of urban wind turbines within the built environment investigating the previously mentioned four points and there effect on the performance of wind turbines in the vicinity of buildings. Parts of the results of the research have been published in several publications and this paper tries to summarise and present all the obtained results in one place [1, 2, 3, 4, 5]. The results focuses on the technology of urban wind turbines, best practice guidelines for using CFD in assessing urban wind flow, effect of wind direction, roof shape, building height and urban configuration on wind flow and the performance of urban wind turbines.

METHODOLOGY
For identifying the appropriate wind turbines technology to be used when installing wind turbines near buildings, literature on the available and developing technologies have been reviewed discussing different types of integration of wind turbines near buildings. Wind turbines have different types based on their technology and the way in which they are integrated within the built environment. Advantages and disadvantages of each type of integration are investigated to determine which type of integration and technology are more relevant to the built environment. Variables affecting wind flow within the built environment are deduced from investigating literature in the field.

For accurately assessing wind resources at the proposed installation site, several wind assessments tools are available. Literature on the available tools for assessing wind flow within the built environment is investigated to identify the advantages and disadvantages of each tool, and accordingly, decide upon the relevant tool to be used in this research. The tool used in this research to assess wind flow above the investigated cases is the CFD code Fluent 12.1 which is used as the experimentation tool to generate data for statistical analysis. However, as CFD simulations are approximations of the real scenarios, they have to be validated by comparing the results with the results of another wind assessment tool. Thus, a detailed validation study is carried out.

After validating the CFD simulation results, the simulation conditions used for the validation study are used for assessing wind flow above six different roof shapes covering a six meters cube isolated building. Flow characteristics are assessed through plotting the flow patterns around the studied shapes and measuring the turbulence intensities and streamwise velocities and normalizing them at different locations above the roofs. The collected data is parametrically analysed and compared to each other to determine the effect of different roof shapes on wind flow above them under different wind directions.

Simulations are carried out to identify the optimum mounting location above the investigated roof shapes. Comparing the results, the optimum roof shape for roof mounting wind turbines is identified. The optimum roof shape is then investigated further by covering isolated buildings of different heights to identify the effect of height on wind flow above the designated roof shape. Since the hypothetical isolated building scenario is not the most commonly encountered scenario within reality, the investigation is carried further to include assessing wind flow above the optimum roof shape covering different buildings' heights placed within different urban configurations.

Accordingly, the effects of height and urban configurations on wind flow above the investigated roof shape are identified. These results are interpreted in terms of energy yield of installed wind turbines to determine the feasibility of the accelerating effect of different roof shapes. Thus, the increase in the wind velocity is transferred into an increase in wind energy to identify the potential increase in energy yield for the proposed roof mounted wind turbine.

URBAN WIND TURBINES TECHNOLOGY

In light of the reviewed literature, it was found that both the mean wind velocity and the turbulence intensity at the installation site are the main factors affecting the energy yield of the wind turbines. In terms of turbine technology it can be argued that for the integration of a turbine within the built environment to be successful, it is recommended:

- Using vertical axis wind turbine (VAWT) to cope with the high levels of turbulence. However, it should always be noted that VAWT have lower power coefficient than HAWTs which will have an impact on the energy output.
- Blades implementing lift forces are more preferable than drag type blades since the first tend to have more power coefficient.
- Latest technology should be implemented; for example the contra-rotating rotating wind turbines system and the concept of a smart wind turbine by Sharma and Madawala [6] which have adjustable blades can be implemented to operate the turbine and generate electricity even at relatively low wind speeds.
- An active yaw system like the one proposed by Wu and Wang [7] would make the wind turbine yield more electricity than self-driven yawing system.
- Using an induction, permanent magnet generator rather than a synchronous generator implementing an electromagnet.

As for understanding wind flow within the built environment, in order to decide about the optimum possible location for urban wind turbines, it is recommended to:

- Mount wind turbines on top of high rise buildings; 30-50% higher than the surrounding urban context.
- Avoid areas with high roughness length where surrounding buildings have the same height as the proposed mounting location.
- Avoid areas with high levels of turbulence or areas around buildings where flow separation occurs. But it should be noted that these areas have high energy content and VAWT can be used.
- Take advantage of the accelerating effect of buildings on wind.
- Place wind turbines between building and preferably buildings with diverging configurations.

It should be noted that a complete wind assessment should take place at the proposed site to understand wind flow at the installation location to avoid areas of high levels of turbulence and choose areas with wind speed relevant to the rated wind speed of the proposed wind turbine. Different tools are available for assessing wind flow at the installation site. These tools include in-situ measurements, wind tunnel tests and computational fluid dynamics (CFD).

CFD FOR ASSESSING URBAN WIND FLOW

Among the available wind assessment tools, it can be argued that CFD simulation is the most relevant tool for implementation in this research since CFD is the most relevant tool for comparing design alternatives and this research mainly focuses on comparing alternative roof shapes and their effect on the energy yield and positioning of roof mounted wind turbines. Thus, CFD as a tool for investigating urban wind flow was investigated further to reach a conclusion about the requirements for a consistent CFD simulation through investigating the main potentials and constrains of using different CFD simulation parameters for assessing wind flow around buildings.

The set of requirements for a consistent CFD simulation is strongly dependant on the availability of adequate computational power and availability of experimental data for validation purposes. Although DNS, LES, DES and URNAS methods yield more reliable results, their implementation in studying wind flow around buildings is few when compared to Steady RANS models. Accordingly, there is a lack in literature for detailed validation for these methods. This is not the case for RANS models where many guidelines and best practice documents can be found in literature. Recommendations regarding the best practice guidelines for implementing CFD in assessing urban wind flow include:

- Second order schemes or above should be used for solving the algebraic equations.
- The scaled residuals should be in the range of 10^{-4} to 10^{-6}.
- Multi-block structured meshes are preferable and carrying out sensitivity analysis with three levels of refinements where the ratio of cells for two consecutive grids should be at least 3.4.
- Mesh cells to be equidistant while refining the mesh in areas of complex flow phenomena.
- If cells are stretched, a ratio not exceeding 1.3 between two consecutive cells should be maintained.
- For flows around isolated buildings, the realizable k-ε turbulence model is preferred.
- Accuracy of the studied buildings should include details of dimension equal to or more than 1 m.
- If H is the height of the highest building the lateral dimension = 2H + Building width, Flow direction dimension = 20H + Building dimension in flow direction and Vertical Direction = 6H while maintaining a blockage ratio below 3 %.
- For the boundary conditions, the bottom would be a non-slip wall with standard wall functions, top and side would be symmetry, outflow would be pressure outlet and inflow would be a log law atmospheric boundary layer profile which should be maintained throughout the length of the domain when it is empty.
- Horizontal homogeneity of ABL profile throughout the computational domain.

Although it is argued that these requirements would lead to a high quality CFD simulation, it is mandatory to validate the CFD simulation using another wind assessment tool to minimise the errors and uncertainties in the CFD code. It can be argued that implementing these parameters in studying wind flow around a 3D cube immersed in a turbulent channel flow would be adequate for validating the CFD simulation results in this research. This would be done by comparing the results with the data sets from published researches investigating wind flow around a cube in a turbulent channel flow.

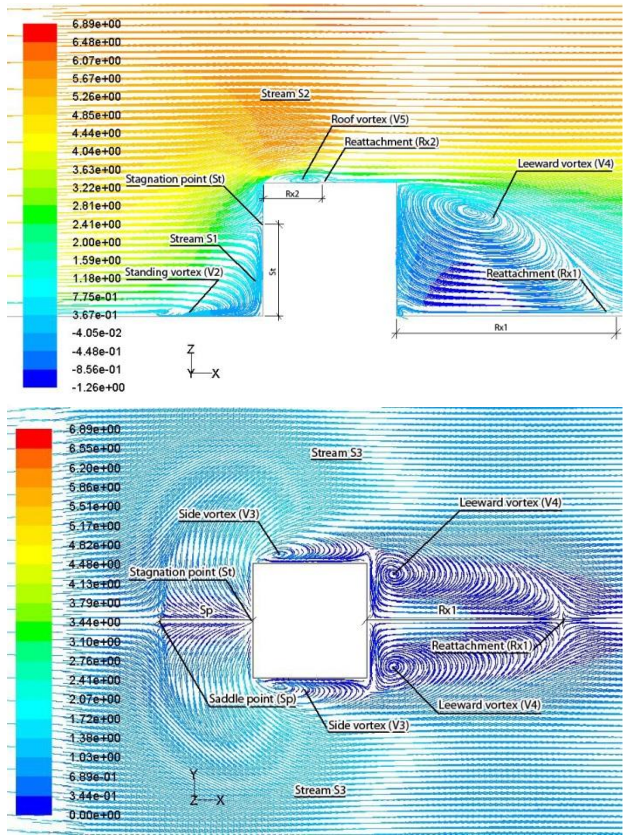

Figure 1 Streamwise velocity pathlines passing through the vertical central plan (top) and ground level (bottom).

Results for this flow problem were compared to published results from in-situ measurements, wind tunnel tests and validated CFD simulations. Qualitatively and quantitatively the results are consistent and compares favourably with other reviewed results as all the flow features were captured in the CFD simulation (Fig. 1). In addition, all the values of the specific lengths of the flow were within the ranges of the reviewed results (Table 1). In general, the results are closest to the wind tunnel results from Castro and Robins [8]. The highest discrepancies were found on the roof in terms of the distribution of maximum pressure coefficients although the values were acceptable and the locations were also acceptable but for the values near the windward edge of the roof some discrepancies were observed. However, these discrepancies where consistently reported in similar published CFD simulations [9, 10] which suggests that the source of the error is numerical. However, when comparing the obtained results with the reviewed validated results and the results from other wind assessment tools, the obtained results compare more favourably than the reviewed CFD simulation results. Accordingly, the used simulations variables can be used in confidence for simulating urban wind flow.

Table 1 specific lengths of the flow around the cube

Sp	St	Rx_2	Rx_1	CpW	CpR	CpL
0.80h	0.81h	0.32h	1.60h	0.81	-0.97	-0.17

Where Sp is the saddle point, St is the stagnation point, Rx_2 is the roof reattachment length, Rx_1 is the reattachment length in the leeward direction of the cube, CpW is the windward maximum positive pressure coefficient, CpR is the maximum negative pressure on top of the roof and CpL is the maximum negative pressure on the leeward façade (CpL).

INVESTIGATED INDEPENDENT VARIABLES

In order to specify the optimum roof shape for mounting wind turbines, the CFD commercial code Fluent 12.1 was used to simulate wind flow above six different roof shapes covering a cubic building whose edge height is six meters, these roof shapes are flat, domed, gabled, pyramidal, barrel vaulted and wedged roofs. To investigate the effect of wind direction, simulations were undertaken with different wind directions; with direction 0^0 is perpendicular to the main axis of the building (parallel to the roof profile).

To understand the effect of each roof shape on air flow above it, streamwise velocity pathlines are plotted along the central plan parallel to the wind direction and to determine the optimum location for mounting a wind turbine on top of each roof, both the turbulence intensity and streamwise velocity were plotted along different locations above each roof extending from directly above roof to a height of 1.5H (H = Height of the of the 6m cube). All roof shapes cover a building of square cross section 6m x 6m and height 6m (Fig. 2). Accordingly, the optimum mounting location for each roof type under different wind directions is specified and the results are compared to each other to determine the optimum roof shape for mounting wind turbines.

In order to investigate the effect of building height on wind flow above the building, the optimum roof shape is used to cover the same building but increasing the height to reach 12m then 24m respectively. Then the three cases (6m, 12m and 24m) are compared to each other in terms of flow pattern, turbulence intensity and streamwise velocity and the effect of height is identified.

For investigating the effect of urban configuration and height on wind flow above the roof of a building, the optimum roof shape is used to cover a 4.5m, 6m, 12m and 24m building placed in an array of cubic buildings whose edge height is 6m, the cubes are arranged in an urban canyon configuration and another time in a staggered urban configuration. Then all the results are compared to each other to identify the effect of roof shape, wind direction, building height and urban configuration on the mounting location and the energy yield of roof mounted wind turbines. The location of the mounting position is given by a grid superimposed onto the roof plan and is shown in Fig. 2.

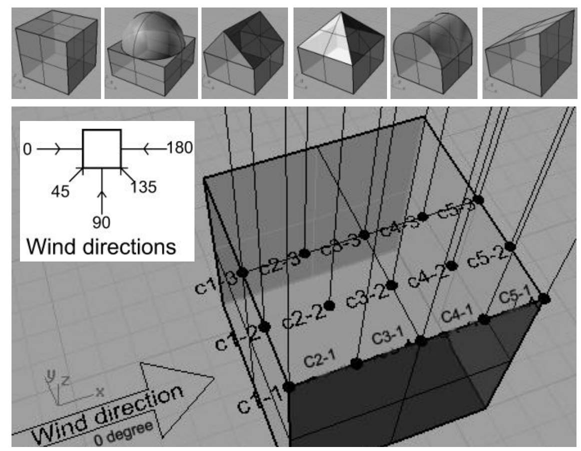

Figure 2 Top: Different investigated roof shapes, Bottom: Locations of measurements and wind directions

For all roof shapes the maximum turbulence intensities were recorded on top of the roof, directly above the roof to a height above the roof of 1.3H. This area should be avoided when mounting wind turbines on the top of these roof shapes. When analysing the streamwise velocities, values in the zone of maximum turbulence intensity were ignored, hence only maximum streamwise velocities above 1.3H were considered.

Table 1 show that all roof shapes caused an increase in the streamwise velocity, with the dome and the barrel vault having the potential to produce significantly more energy than the other roof shapes. Since the energy yield of a wind turbine is directly proportional to cube the wind speed, the domed roof would yield 40.5% increase in power whilst the barrel vaulted roof would yield 56.1% increase in energy. For these two shapes, these maximums were achieved above the centre of the roofs.

For wind directions other than perpendicular to the main axis of the building, symmetry of the roof shapes simplifies the analysis. For the flat, domed and pyramidal roofs, only two directions were modelled; perpendicular and at 45^o. The wind directions for the gabled and barrel vaulted roof were perpendicular, 45^o and 90^o, all other directions being symmetrical. Only the wedge roof shape was modelled in all wind directions.

Table 1: Maximum normalised velocities and locations for wind direction perpendicular to the roof shape.

Roof Shape	Maximum Normalised velocity	Location
Flat	1.095	2-3 at 1.45H
Domed	1.12	3-3 at 1.3H
Gabled	1.05	5-1 at 1.6H
Pyramidal	1.05	4-2 at 1.3H
Barrel Vaulted	1.16	3-3 at 1.3H
Wedged	1.03	5-1 at 1.45H

For a wind direction of 45^{o} for the flat, domed and pyramidal roof shapes, the maximum normalised velocity were 1.12, 1.14 and 1.08 and the locations were 2-2, 3-3 and 4-4 respectively. The gabled and barrel vaulted roof, for a wind direction of 45^{o} had maximum normalised velocities of 1.09 and 1.14 at locations 3-5 and 3-3 respectively. And for the 90^{o} wind direction, here the wind direction is perpendicular to the gable ends; the maximum normalised velocities were 1.075 and 1.083 at locations 2-3 and 2-2 respectively. For the wedge shaped roof the maximum normalised velocities at 45^{o}, 90^{o} 135^{o} and 180^{o} were 1.07, 1.075, 1.14, 1.08, at locations 5-5, 2-4, 3-2, 2-1 respectively.

Having established the performance of the roof shapes for a 6 metre high building, further simulations were undertaken to investigate how the building's height changes the roof performance in terms of accelerating wind. As the barrel vaulted roof, with the wind direction perpendicular to the vault had the highest increased wind velocities above it, this was chosen as the test case to investigate the influence of building height. Two building heights were modelled; one of 12 metres and the other of 24 metres, both are compared to the 6 metre case.

In all three cases the maximum turbulence intensity and maximum streamwise velocity occurred at the same locations. However, there was an increase in the turbulence intensity with increase in the height of the building, which suggests that there is a relationship between the building height and the turbulence intensity. The normalised streamwise velocities for the 6 metre and 12 metre cases at 1.3H were similar, whilst the 24 metre case gave a value of 1.175 at 1.3H. This shows that relationship between building height and the ground starts to become less significant

The results presented so far have been for an isolated building, with the wind accelerating around the building, a rural scenario. To start to understand the performance of roof shapes in an urban context, a final series of simulations were undertaken within two different urban configurations. These two configurations were the street canyon and the staggered street, the former representing an ordered layout of the urban environment and the latter a more chaotic configuration. Fig. 3 shows the staggered urban configuration, in the centre of these configurations a barrel vaulted building is inserted, whose height was altered between 4.5 metres, 6 metres, 12 metres and 24 metres. The surrounding buildings were kept at a fixed height of 6 metres. Wind direction was as perpendicular to the main axis of the barrel vaulted roof (parallel to the roof profile), this wind direction had provided the maximum streamwise velocity over the isolated roof shape.

The results from these last series of simulations need to be split into two groups to be understood, the first group is where the modelled building is below or equal its surroundings and the second group where the modelled building is above the surroundings. In first group the surface roughness dominates the air flow, whilst in the second group the roughness effect is less marked.

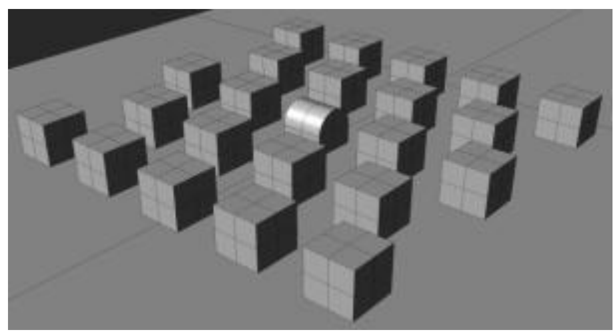

Figure 3: The two urban configurations modelled, top the street canyon and the bottom the staggered street, both show the barrelled vaulted building in the centre at a height of 6 metres

For the 4.5 metre case, the turbulence intensity diminishes above 1.6H, and the maximum normalised streamwise velocities reach a maximum at 2.5H. At this height the staggered configuration has a normalised streamvelocity of 1.09, whilst the canyon configuration value is 1.07. When the barrelled vault is at the same height as its surroundings, the roughness of the surroundings increases the turbulence and pushes the position of the maximum normalised streamwise velocity to 2.5H. The values and order were identical to the 4.5 metre case.

As the barrel vaulted building rises above the surroundings (the second group) the roughness effects diminish, resulting in the turbulence intensity and normalised streamwise velocity profiles corresponding to the isolated building case. For both building heights and urban configurations, the turbulence intensity becomes less significant above 1.3H, following the performance of the isolated building. For the 12 metres high case the canyon and staggered configuration have a normalised streamwise velocities of 1.13 and 1.1 respectively at 1.3H. And the 24 metre height case has normalised streamwise velocities of 1.15 and 1.13 for the canyon and staggered configurations respectively. A reversal of the situation found in first group in that the canyon configuration has less impact on velocity than the staggered configuration, but for both cases the normalised streamwise velocities were less than the isolated building case.

CONCLUSION

Urban wind turbines is a relatively new field which is developing and has high potentials with the advancements in small and micro scale wind turbines technologies and the continues investigation of taking advantage of the accelerating effect of different buildings' shapes. This research goes some way towards addressing the developing wind turbines technologies to be integrated within buildings in addition to investigating the accelerating effect of different roof shapes. However, from a practical and architectural point of view, how can the results of this research be implemented?

In order to answer this question, it should be noted that the idea of integrating wind turbines in urban areas is still questionable due to the low mean wind speed and high levels of turbulence in addition to the difficulty of assessing, to a high degree of accuracy, the wind resources at the proposed mounting location. However, when it comes to mounting wind turbines near buildings in rural areas or on top of isolated buildings' roofs in open fields, the integrated wind turbines would have more potential in terms of being mounted at the optimum mounting location to take advantage of the accelerating effect of the building.

These areas are usually located away from the grid and energy consumption is minimal which makes the idea of integrating renewables more viable. For roof mounting wind turbines, the case will either be retrofitting an existing building with a roof mounted wind turbine or a new building is being built and the decision has been made to rely on wind energy as part of energy supply of the building. In the first case, the wind resources can be assessed to a high degree of accuracy due to the simplicity of the surrounding context. Thus, the optimum mounting location can be determined. However, the structural integrity of the building and any other potential problems from retrofitting the existing building with the wind turbine should always be assessed before installing the wind turbine.

As for the second case and in light of the obtained results in this research, a recommendation can be made to the developer on which roof shape to be used and how to orient the building in a way in which the roof mounted wind turbine could benefit from the prevailing wind direction and its interaction with the proposed roof shape.

Accordingly, the optimum mounting location can be determined and the anticipated energy yield can be calculated before the inception of the project which will help in deciding about the feasibility of roof mounting a wind turbine. However, it should always be noted that there will be a compromise between the orientation of the building, the roof shape and other architectural requirements whether being ecological, functional or even certain specific requirements by the developer. But it can be argued that such integration can result in a new type of buildings where the form of the building will follow its function from a power related point of view.

REFERENCES

1. Abohela, I., Hamza, N. and Dudek, S. (2011a) 'Assessment of wind flow within the built environment', *Built and Natural Environment*, pp. 81-94.

2.Abohela, I., Hamza, N. and Dudek, S. (2011b) 'Urban Wind Turbines Integration in the Built Form and Environment', *FORUM Ejournal for Postgraduate Studies in Architecture, Planning and Landscape*, 10(1), pp. 23-39.

3.Abohela, I., Hamza, N. and Dudek, S. (2012a) 'Effect of roof shape, building height and urban configuration on the sitting location of roof mounted wind turbines', *PLEA2012 - 28th Conference, Opportunities, Limits & Needs Towards an environmentally responsible architecture* Lima, Perú, 7-9 November 2012. p. Under publication.

4.Abohela, I., Hamza, N. and Dudek, S. (2012b) 'Validating CFD simulations of wind flow around a surface mounted cube in a turbulent channel flow', *PLEA2012 - 28th Conference, Opportunities, Limits & Needs Towards an environmentally responsible architecture* Lima, Perú, 7-9 November 2012. p. Under publication.

5.Abohela, I., Hamza, N. and Dudek, S. (2013) 'Effect of roof shape, wind direction, building height and urban configuration on the energy yield and positioning of roof mounted wind turbines', *Renewable Energy*, 50(0), pp. 1106-1118.

6.Sharma, R.N. and Madawala, U.K. (2012) 'The concept of a smart wind turbine system', *Renewable Energy*, 39(1), pp. 403-410.

7.Wu, Z. and Wang, H. (2012) 'Research on Active Yaw Mechanism of Small Wind Turbines', *Energy Procedia*, 16, Part A(0), pp. 53-57.

8.Castro, I.P. and Robins, A.G. (1977) 'The flow around a surface-mounted cube in uniform and turbulent streams', *Journal of Fluid Mechanics*, vol. 79, pp. 307-335.

9.Cóstola, D., Blocken, B. and Hensen, J. L. M. (2009) 'Overview of pressure coefficient data in building energy simulation and airflow network programs', *Building and Environment*, 44(10), pp. 2027-2036.

10.Franke, J. (2007) 'Introduction to the Prediction of Wind Loads on Buildings by Computational Wind Engineering (CWE)', in Stathopoulos, T. and Baniotopoulos, C. C. (eds.) *Wind Effects on Buildings and Design of Wind-Sensitive Structures*. Springer Vienna, pp. 67-103.